

Users Guide for the Meshparts Software

1 System Requirements

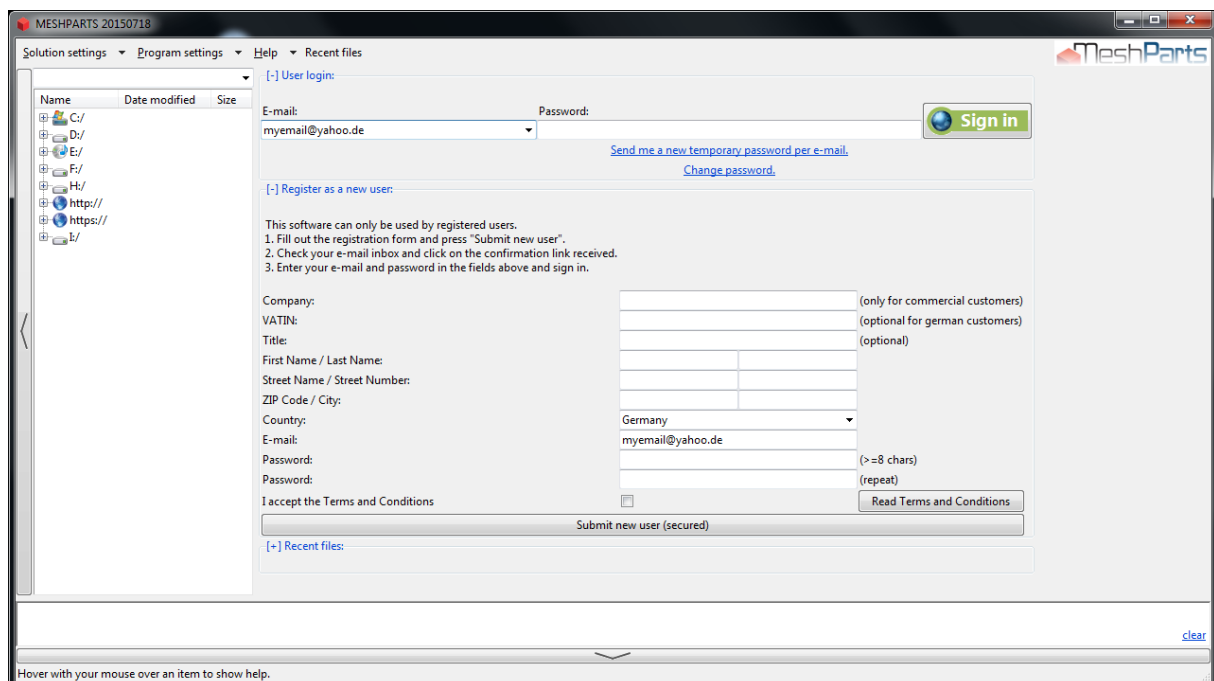
The MESHPARTS Software in the current version runs only on Windows operating systems. The software was tested on Windows XP and Windows 7 (64 Bit and 32 Bit). There is no special requirement regarding the memory amount and processor speed but with bigger finite element models more memory will be needed.

2 Download

The software can be downloaded from www.meshparts.de/software

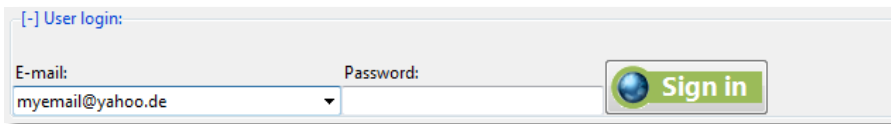
3 Quick start guide

1. Go to www.meshparts.de/software and download the latest release of the MESHPARTS Software.
2. Save the executable on your hard drive (preferable on C:\, D:\ or other short paths) and run it **as an administrator** (Windows context menu for executable files). On a Windows 7 operating system the graphical user interface (GUI) looks like this:



3. Click on “Register as a new user” and create a new account by registering with your e-mail address and a password (additional information required). After submitting your data, you will receive a confirmation e-mail. Click on the link in the e-mail.

4. Login with your e-mail address and password (same as used for registering on step 3):



[-] User login:

E-mail: myemail@yahoo.de Password:



5. Check the server response in the message window. You should see a similar message and a question to setup the offline library:

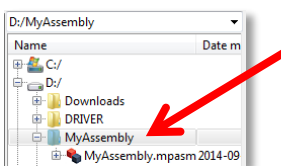
```
*** Important action required: Do you want to set the offline library path for https://www.meshparts.de now? Yes or No
Login successful. License granted.
Licence type: commercial; Software updates and technical support included until 2015-December-19.
Remaining licences: 9
```

6. If you log in for the first time and the offline library is not set yet, you should click on “Yes” in order to set the offline path for www.meshpart.de, as this step is very important for the proper functioning of the software.

4 Your first Finite Element Assembly

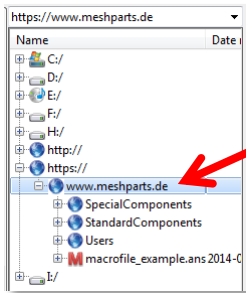
The MESHPARTS Online Library of parametrical finite element models is directly accessible from the MESHPARTS Software. This allows users to create directly finite element assemblies without having a local geometry meshing software installed.

1. In the explorer tree right click on a folder of your choice and choose from the context menu **New assembly**. The new assembly opens automatically.

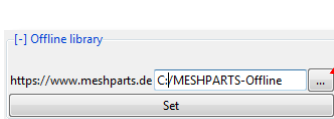


If you already set up the path to the offline library as described in Chapter 3, you can skip the steps 2 and 3.

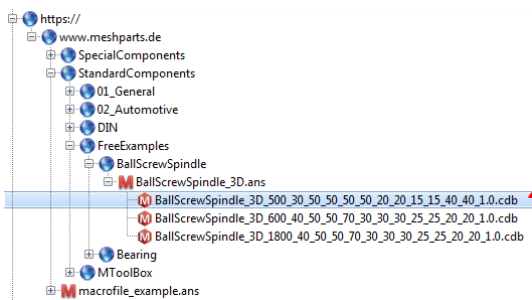
2. Select from the explorer tree the path <https://www.meshparts.de>



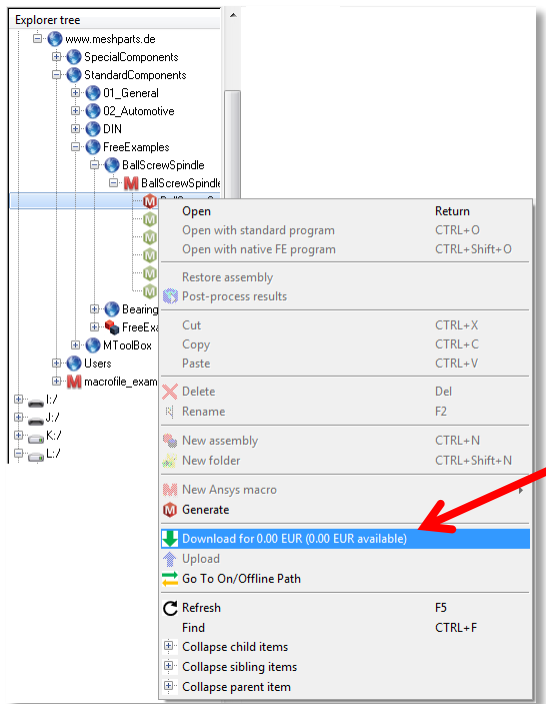
3. In the Offline Library frame, set the path to your **offline library** (browse to the path and click on "OK"). All models downloaded from the online library will be mirrored to this path.



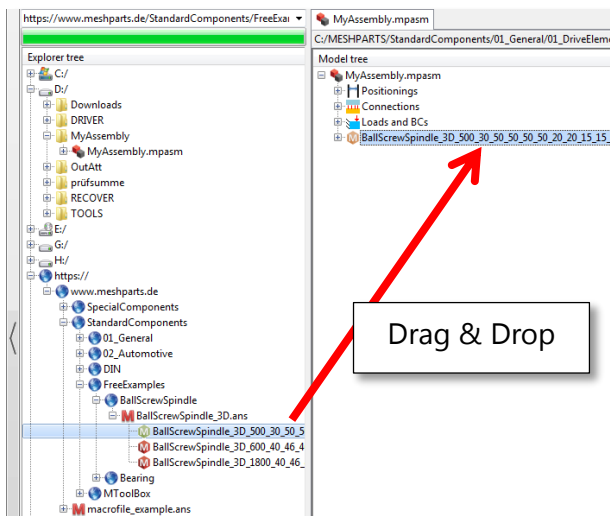
4. **Browse** in the explorer tree to https://www.meshparts.de/StandardComponents/FreeExamples/BallScrewSpindle/BallScrewSpindle_3D_500_30_50_50_50_20_20_15_15_40_40_1.0.cdb (you can also copy and paste the path into the path field above the explorer tree)



5. Right click the model and select "**Download**"



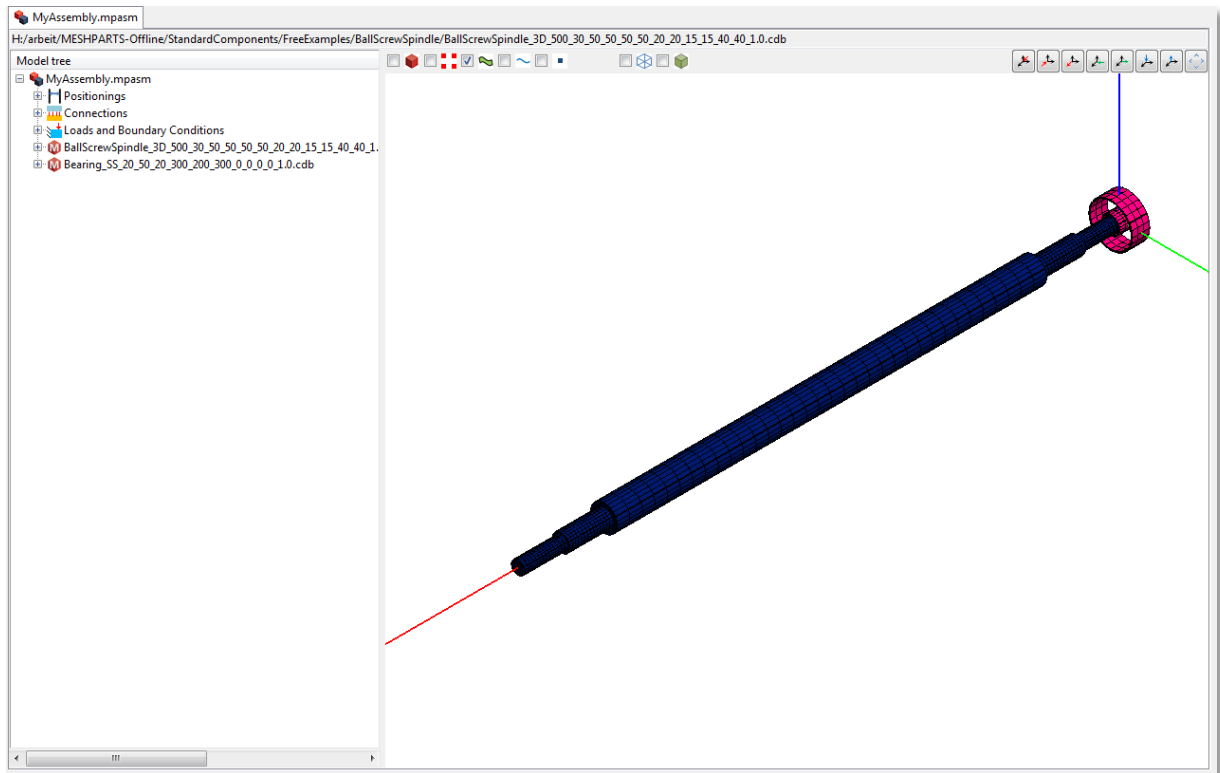
6. **Drag and drop** the downloaded model to the model tree of the new assembly.



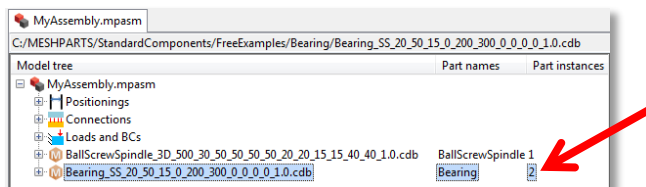
7. Repeat steps 4, 5 and 6 for the model

https://www.meshparts.de/StandardComponents/FreeExamples/Bearing/Bearing_SS_20_50_15_0_200_300_0_0_0_0_1.0.cdb.

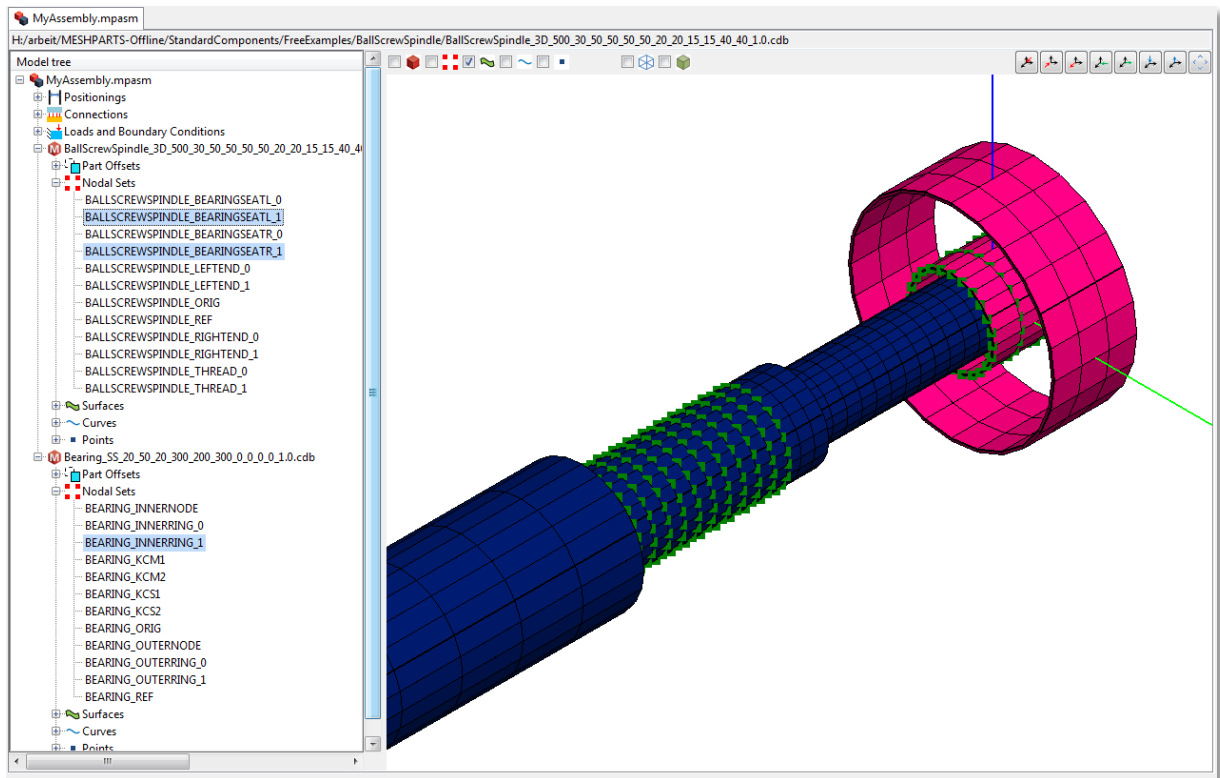
Your new assembly looks like this:



8. Scroll to the right in the model tree and change the number of **part instances** of the bearing from 1 to 2 by choosing "Rename" from the context menu.



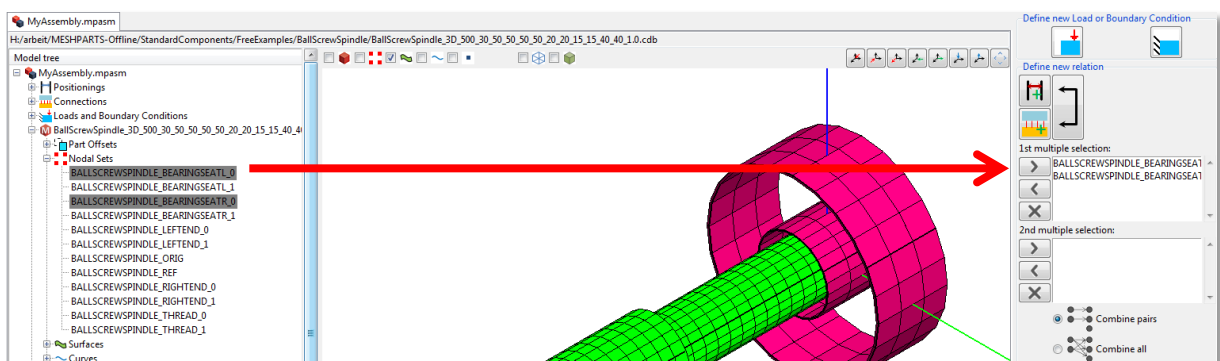
9. From the model tree **select the nodal sets** BALLSCREWSPINDLE_BEARINGSEATL_1, BALLSCREWSPINDLE_BEARINGSEATR_1 and BEARING_INNERRING_1. Alternatively, you can also select surfaces directly on the 3D model but selecting nodal sets is more robust with respect to model changes. Hold down the **control key** in order to make a multiple selection.



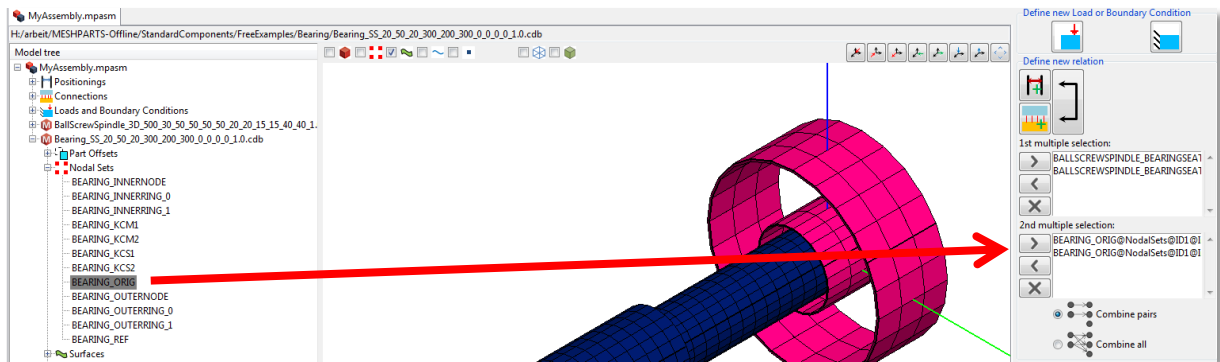
10. With one click on the second button in the "Define new relation" frame, define a new **contact relation** between the selected nodal sets.



11. From the model tree **select the nodal sets** BALLSCREWSPINDLE_BEARINGSEATL_0, BALLSCREWSPINDLE_BEARINGSEATR_0 and click on the right arrow in the "1st multiple selection" box:



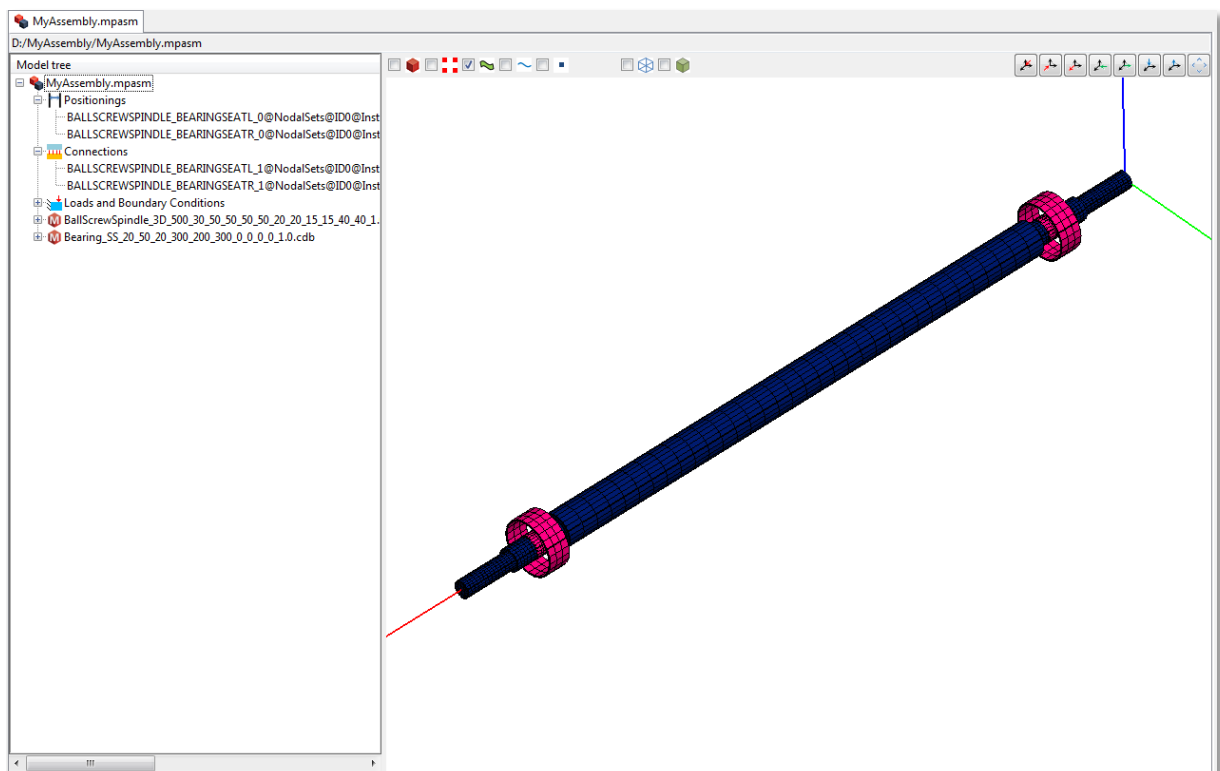
12. From the model tree **select the nodal set** BEARING_ORIG and click on the right arrow in the "2nd multiple selection" box:



13. With one click on the first button in the "Define new relation" frame, define a new **positioning relation** between the selected nodal sets.



Your assembly is now fully defined and looks like this:

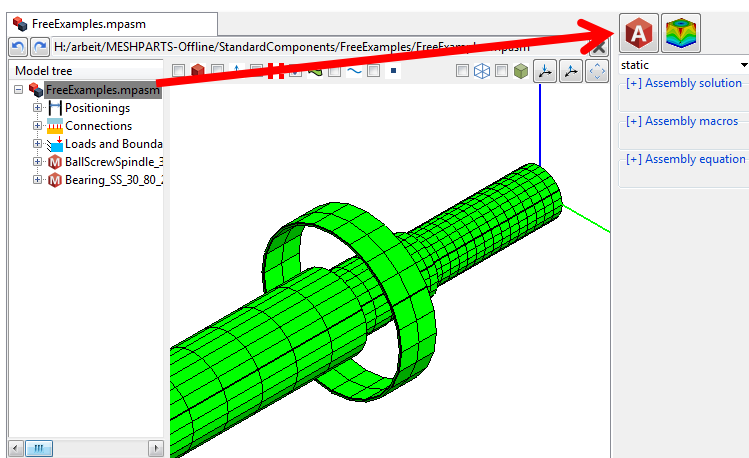


5 Exporting Finite Element Assemblies

MESHPARTS finite element assemblies are exported (converted) into a file format that can be read by third-party finite element software. Currently the Ansys® specific CDB file format is supported. Ansys® CDB models can also be imported into Abaqus®.

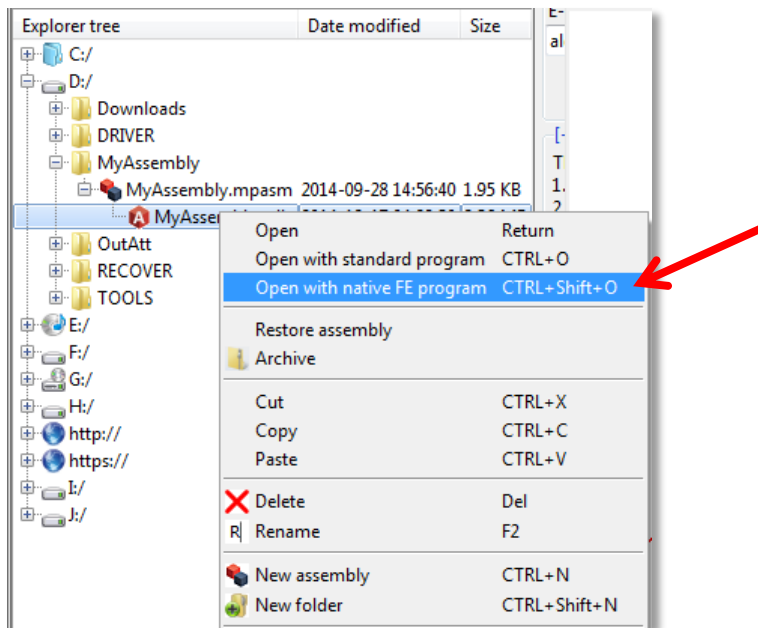
In most cases, you will want to open the MESHPARTS assemblies with a third-party finite element software in order to post-process the finite element results. The pre-processing and the solution setup you can perform directly in the MESHPARTS software.

1. In order to export MESHPARTS assemblies to Ansys® CDB format select an assembly item in a model tree and click on the button :



The new exported model will appear as a child item of the assembly file in the explorer tree (click on the plus sign if necessary).

2. Right click on the exported model and choose "Open with native FE program". The model opens in Ansys® Workbench or Mechanical APDL (depending on the program settings).



6 Meaning of icons

In order to work with the MESHPARTS Software it is important to know the meaning of the icons used.



This is an operating system folder on your local or network drive.



This is a folder from the online library.



This is a macro file (e.g. Ansys® APDL macro library with MESHPARTS specific structure). From macro files, model files are generated and grouped as child items under the macro item.



An orphan model file (part); the origin of the model file is unknown.



A model file generated from a MESHPARTS macro file.



A model file generated from a MESHPARTS assembly file.



A MESHPARTS assembly file. An assembly file contains definitions of assemblies that can consist of model files or other assembly files.



A macro generated model file that you should re-generate or download, because it is older than the macro file or online version respectively.



An assembly generated model file that you should re-generate or download, because it is older than the assembly file or online version respectively.



An assembly file that you should re-download, because it is older than the online version.



An orphan model file from the online library that is also available in your offline library.



A macro generated model file from the online library that is also available in your offline library.



An assembly generated model file from the online library that is also available in your offline library.



An assembly file from the online library that is also available in your offline library.



A macro generated model file whose macro file is not available anymore.



This is an assembly generated model file whose assembly file is not available anymore.



An orphan model file that is not available anymore.



A macro generated model file that is not available anymore.



An assembly generated model file that is not available anymore.

7 Changing the viewing angle, zooming, and panning

In order to change the viewing angle, zooming, and panning of your models you will need a mouse with three buttons (left, right and middle button) and a scroll wheel. For panning the model view, you will need a keyboard.

- **Change the viewing angle of your model:** Hold down the middle button on your mouse while moving the mouse horizontally and vertically.
- **Zoom in and out:** Turn the mouse wheel up and down.
- **Pan your model horizontally and vertically:** Hold down the control key on your keyboard and the middle button on your mouse while moving the mouse horizontally and vertically.

The actions described above base on the current mouse position relative to the model. Also, consider following two point, when rotating the model:

- If the drag operation begins with the mouse positioned over the model, the rotation point is the intersection point of the viewing direction with the model.
- If the drag operation begins with the mouse positioned beside the model, the rotation point is the center of the model.

7.1 Predefined views

On the upper-right corner of the model viewing area, you can find eight buttons for setting predefined viewing directions and zooming.



From the left to the right following predefined settings are available:

1. Isometric view
2. Front view (negative x axis direction)
3. Back view (negative x axis direction)
4. Right view (negative y axis direction)
5. Left view (negative y axis direction)
6. Top view (negative z axis direction)
7. Bottom view (negative z axis direction)
8. Fit model into the viewing area
9. Restore last exploded view

7.2 Exploded views

When different models in an assembly overlap or share common surfaces, it is more difficult to make selections. A recommended way to overcome this difficulty is to create an exploded view of your assembly.

You can create exploded views very fast by simply dragging away different models or assemblies with the left mouse button (hold down the left mouse button and move the mouse).

Additionally, single parts or assemblies can be rotated by dragging the mouse while holding the right mouse button pressed. Per default, the rotation takes place around the origin of the part or assembly. If you hold down the CTRL key while dragging the mouse, rotation will take place around the selected point on the model.

If you hold down the shift key while dragging the mouse (either left or right button), then the part deepest available in the assembly hierarchy will be translated or rotated. Of you do not use the shift key, then the part or assembly, which is a direct child of the main assembly, will be translated or rotated.

7.3 Wireframe, smooth and section views

Directly over the 3D viewing area, you can find four further buttons that will change the way your model or assembly looks like:



-  Wireframe: Only curves and other selected geometry elements are visible.
-  Smooth surface: The mesh edges are invisible.
-  Translucent surface: The mesh faces are 25% opaque.
-  Transparent surface: The mesh faces are 0% opaque.
-  Section view: The model or assembly is cut with the selected geometry element of type "PLANE".
-  Section view reversed: Same as above, but the planes normal direction is reversed.

8 Selection of geometric entities

There are five types of geometric entities available for selection:

1. Models (e.g. Ansys CDB models)
2. References (model origin, Cartesian axes, Cartesian planes)
3. Nodal Sets (groups of finite element nodes)
4. Surfaces (planes, cylinders, spheres)
5. Curves (lines, circles)
6. Points

The model origin, Cartesian axes and planes are treated as points, lines and planes respectively.

You can select geometric entities by one single left mouse click. Holding down the control key on your keyboard enables you to select multiple geometric entities at the same time. Selected entities change their color to green.

There are two possibilities for selecting geometric entities:

1. Selection from the model tree
2. Direct selection on the model

The selection from the model tree is a very convenient way to select entities if you know their names (e.g. nodal sets).

The direct selection on the model is the best way to select geometric entities if you do not know their names.

In some cases, when the bounding boxes of two or more geometry entities overlap, more than one entity is found by a select operation. In those cases, there are three possibilities in order to choose the right entity:

1. Repeated clicks on the same position will cycle through all selectable entities
2. Right click on the model and choosing a geometric entity from the context menu.
3. Drag with the left mouse button some models away from their original position in order to get better access to the geometry entities you want to select.

8.1 Selection filter

On the upper-left corner of the model viewing area, you can find six buttons for filtering the selection of specific geometric entities.



From the left to the right following filters are available:

1. Model filter (select only models)
2. References filter (select only references)
3. Nodal set filter (select only nodal sets)
4. Surface filter (select only surfaces)
5. Curve filter (select only curves)
6. Point filter (select only points)

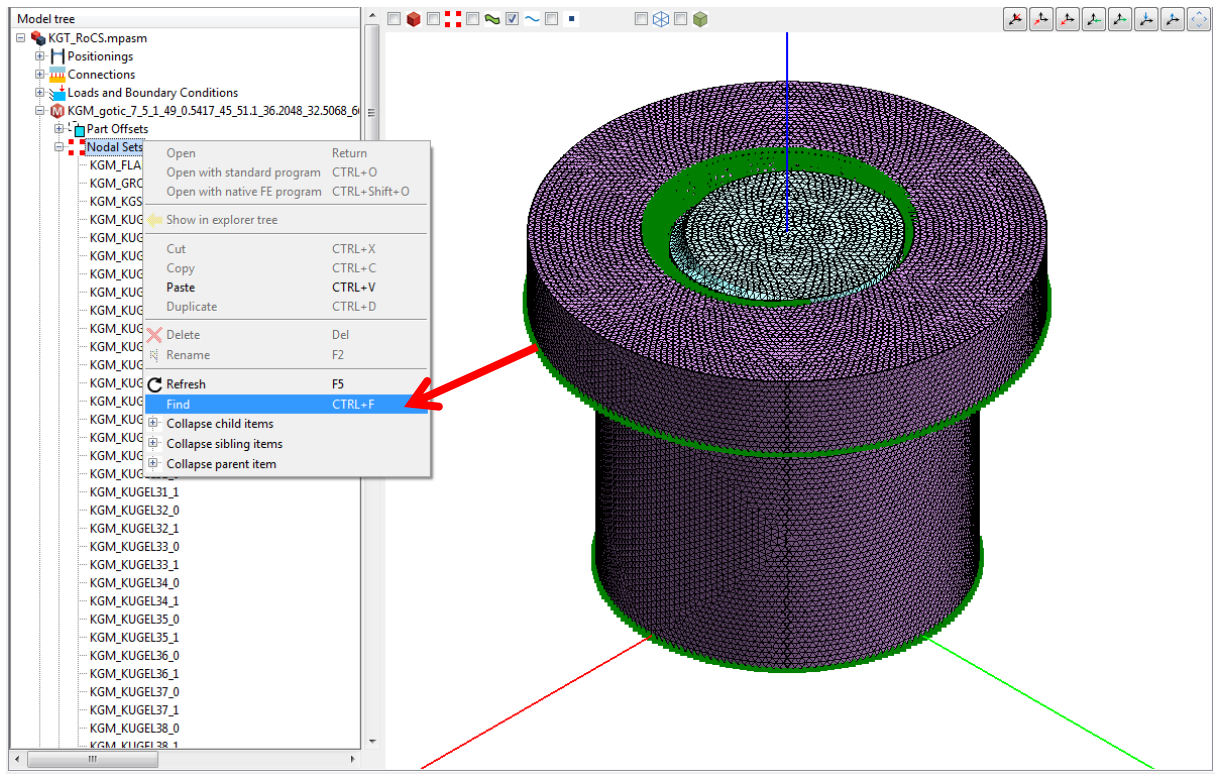
The behavior of the selection filter in the toolbar is additive. This means, that you have to explicitly activate and deactivate each filter option manually.

Another way of changing the selection filter is to show a context menu by a right click on the 3D area. The context menu also shows the corresponding key shortcuts.

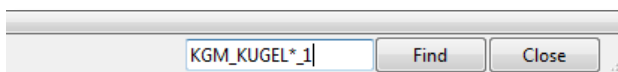
The behavior of the selection filter in the context menu is exclusive. This means, that activating a filter option, will automatically deactivate all other filter options. Nevertheless, if you hold down the CTRL key while choosing a filter option, the filter will behave additive.

8.2 The find function

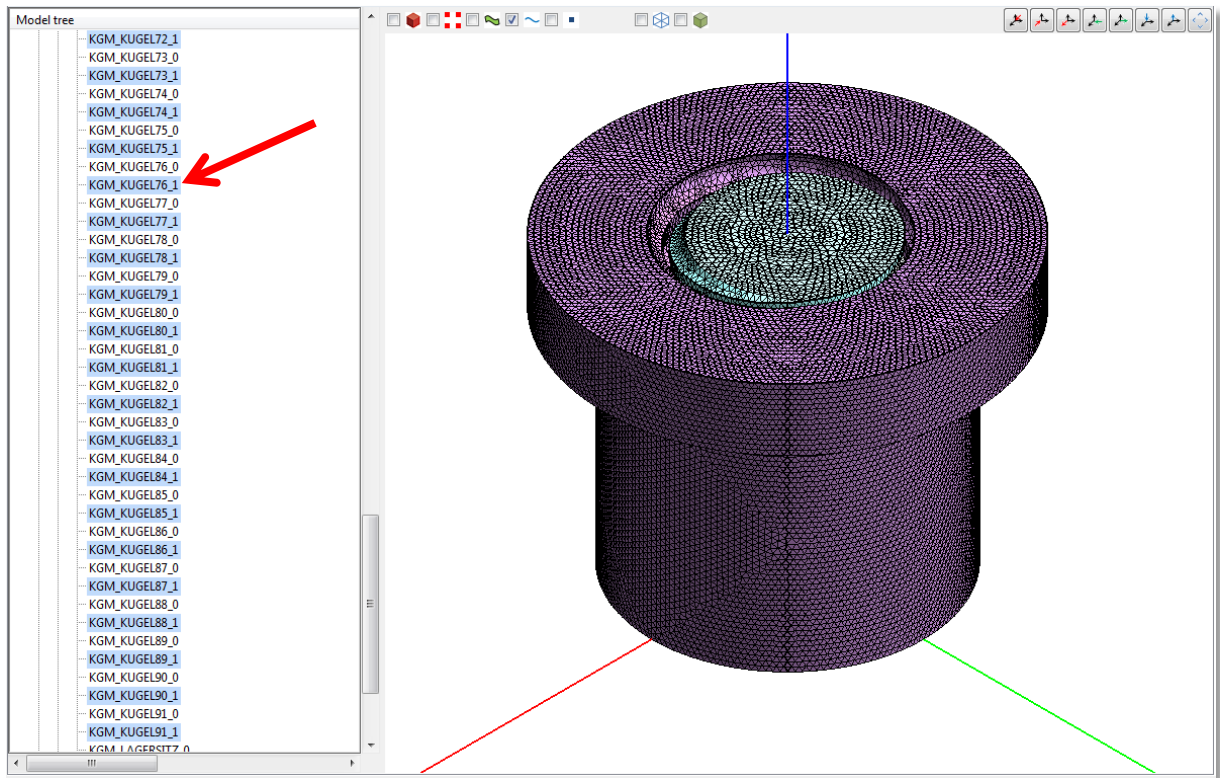
In some cases, you want to select some geometry items that have a specific name. Instead of selecting the items one by one, you can use the Find function from the context menu or by pressing CTRL+F in the model tree:



In the example above, we have the FE assembly of a ball screw drive and we want to select the nodal sets beginning with KGM_KUGEL and ending with _1. We can input the search criteria KGM_KUGEL*_1 (the asterisk is a placeholder for any characters) into the search field that opens when you activate the Find function:



The search starts by pressing Return or by clicking in the "Find" button. The found items are then automatically selected in the model tree.



You can also use the Find function in order to find files or folders in the explorer tree. In that case, make sure that the explorer tree is the active tree by clicking inside it and selecting the folder where you want to start searching.

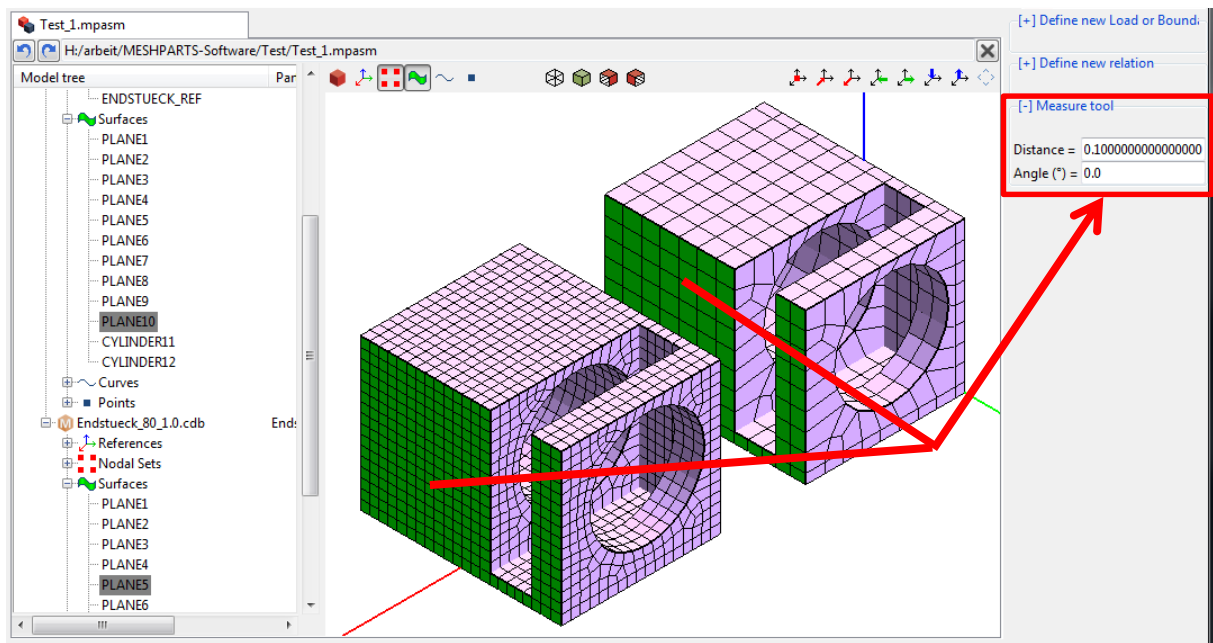
The Find function always refers to the selected folder. If nothing is selected the search is performed over all available folders. Searching over a computer volume or even all available volumes can require up to several minutes of time.

If you press the CTRL key while the Find function is started, the found items are added to the previous selection.

9 Measuring distances and angles

It is a common task to measure the distance or angle between two surfaces or other geometric entities.

In the MESHPARTS software you can select two geometry entities from the model tree or directly on the 3D viewing area in order to measure (if applicable) the distance and angle between them.



Distances are valid for following combinations:

- Point-Point
- Point-Line
- Point-Plane
- Point-Sphere (center)
- Point-Cylinder (middle axis)
- Point-Circle (normal axis through the center)
- Line-Line
- Line-Sphere (center)
- Line-Cylinder (middle axis)
- Line-Circle (normal axis through the center)
- Plane-Plane (if parallel)

Angles valid for following combinations:

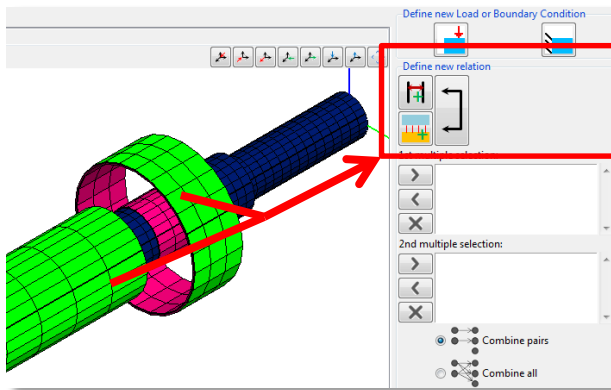
- Line-Line
- Line-Cylinder (middle axis)
- Line-Circle (normal axis through the center)
- Plane-Plane

10 Defining relations between models

A relation between two models can be of two types:

1. Positioning
2. Connection

In order to define a positioning and/or a constraint you must select at least two geometry entities and then click on one of the three buttons in the "Define new relation" frame:



You can also define multiple relations at the same time. For that select multiple geometry items and place the selection into the multiple selection fields.

Choose how to create relations pairs using the two multiple selections:

- The "Combine pairs" option combines the first item in the first multiple selection with the first item in the second multiple selection. The same will apply to the second items and so on. If one multiple selection contains less items than the other one, the creation of pairs stops the last possible pair.
- The "Combine all" option generates all possible combinations of two geometry items. Use this option with caution as it can create a huge amount of new relations.

10.1 Positioning constraints

Positioning constraints are geometric constraints between pairs of models in order to define their relative position. You can select any combination of

- Lines
- Circles
- Planes
- Cylinders
- Spheres

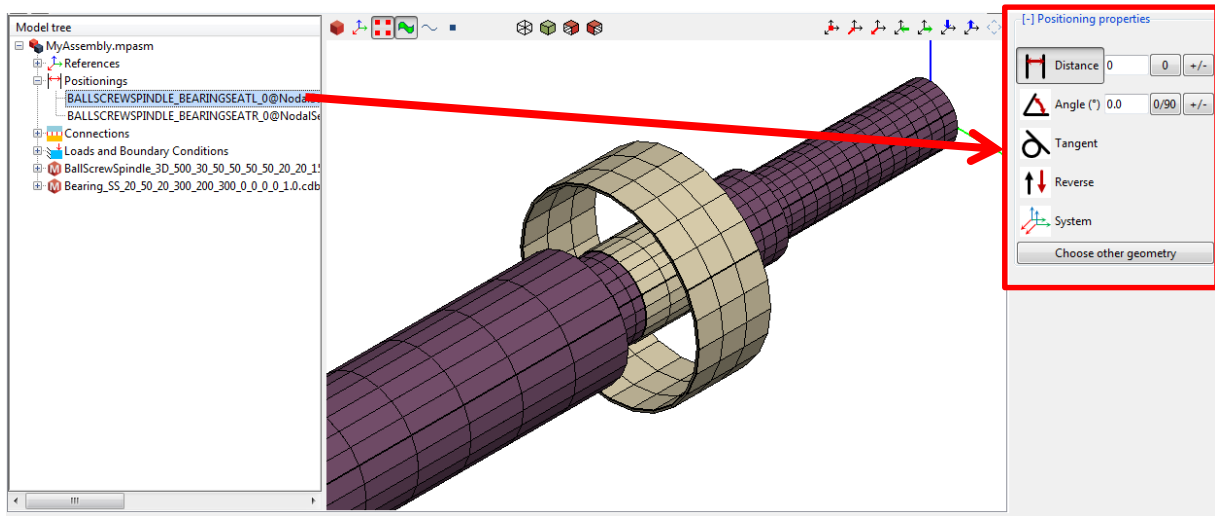
in order to define a positioning constraint.

A positioning constraint can be of three types:

- Distance (in model length units)
- Angle (in degrees)
- Tangent

A distance constraint implies that the angle between the direction vectors of the geometric entities is zero.

The type of a positioning constraint can be set up in the "Positioning properties" frame by selecting one of the positioning items in the model tree:



In addition to the type of a constraint, the direction of a constraint can be reverted with the help of the "Reverse" option.

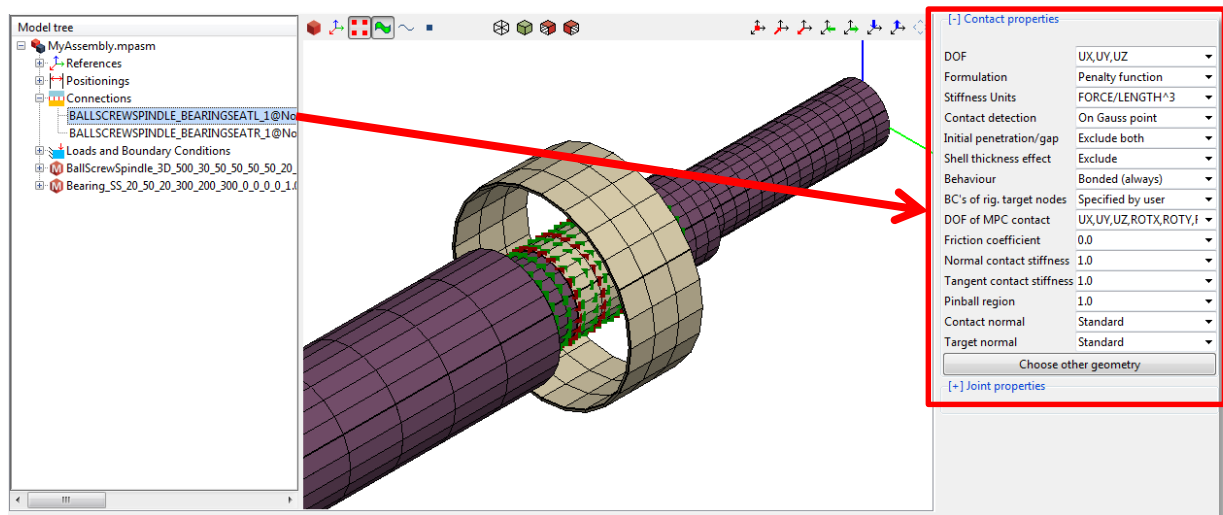
The "System" option is only valid for positioning constraints between two points. This will also constrain the coordinate systems of each point to have the same orientation of their axes.

10.2 Connections

Connections define a physical interaction between two geometry entities (e.g. contact). You can define one of the following connection types:

- surface-surface
- curve-surface
- point-surface
- curve-curve
- point-curve
- point-point

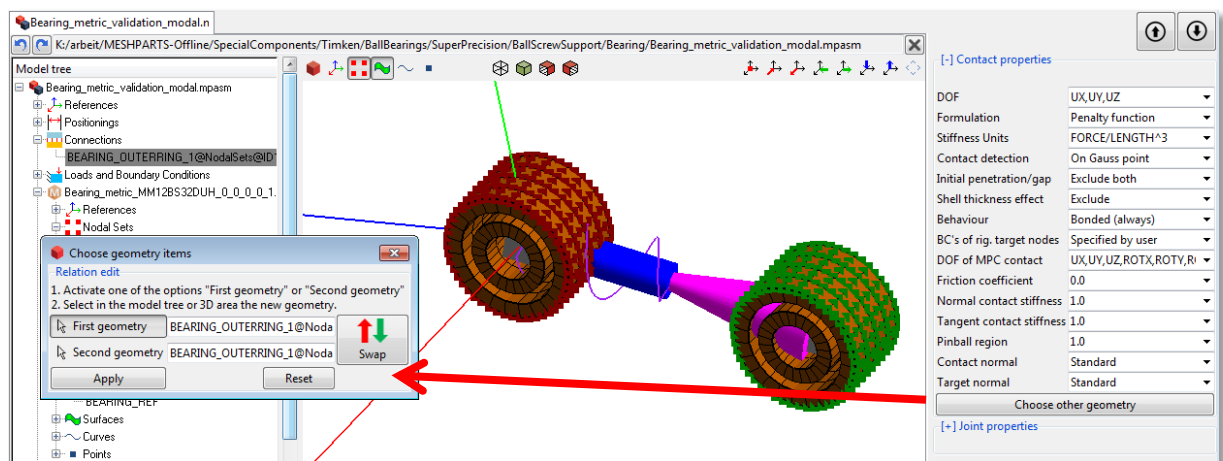
Most of the connections are modeled per default by bounded penalty contact (similar to gluing). Point-point and point-curve connections modeled per default by constraint equations. Other contact options are available when you select a specific connection item in the model tree:



You can read more about each contact option in the Ansys® User's Manual under // Contact Technology Guide // 3. Surface-to-Surface Contact // 3.9. Set the Real Constants and Element KEYOPTS.

10.3 Changing the geometry items in a relation

If you want to change one or both geometry items in a relation (positioning or connection), select the corresponding relation item in the model tree and click on "Choose other geometry" in the relation properties frame on the right side of the window.



11 Applying loads and boundary conditions

You can apply loads and boundary conditions (LBCs) to all type of geometry items available with the exception of reference planes (XY, YZ and ZX items) and reference axes (X, Y, Z items).

On surfaces or equivalent nodal sets, LBCs are applied through external points (pilot nodes) that connect to the surfaces through Multi Point Constraints (MPC).

On curves and points or equivalent nodal sets, LBCs act on every single node of the mesh associated with the geometry. Please take into account, that nodal loads and nodal BCs are defined in the node coordinate system. In MESHPARTS if a model is rotated then also the node coordinate system are rotated and so the LBCs.

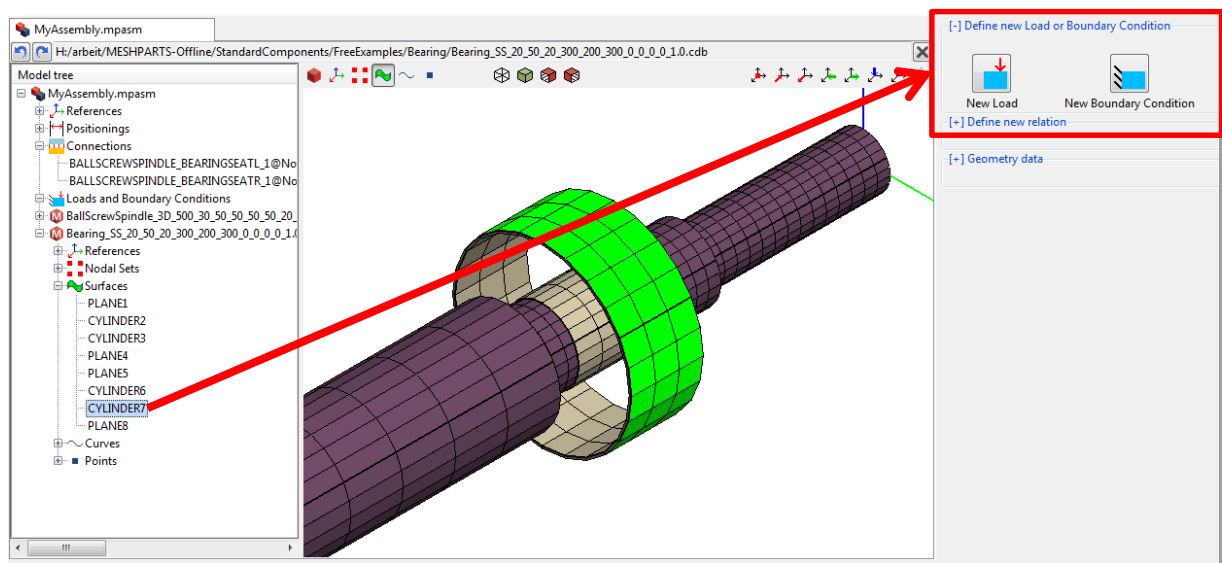
On Origin items you can apply loads of following types in the global coordinate system:

- Linear acceleration
- Rotational acceleration
- Linear velocity (as initial condition)
- Rotational velocity

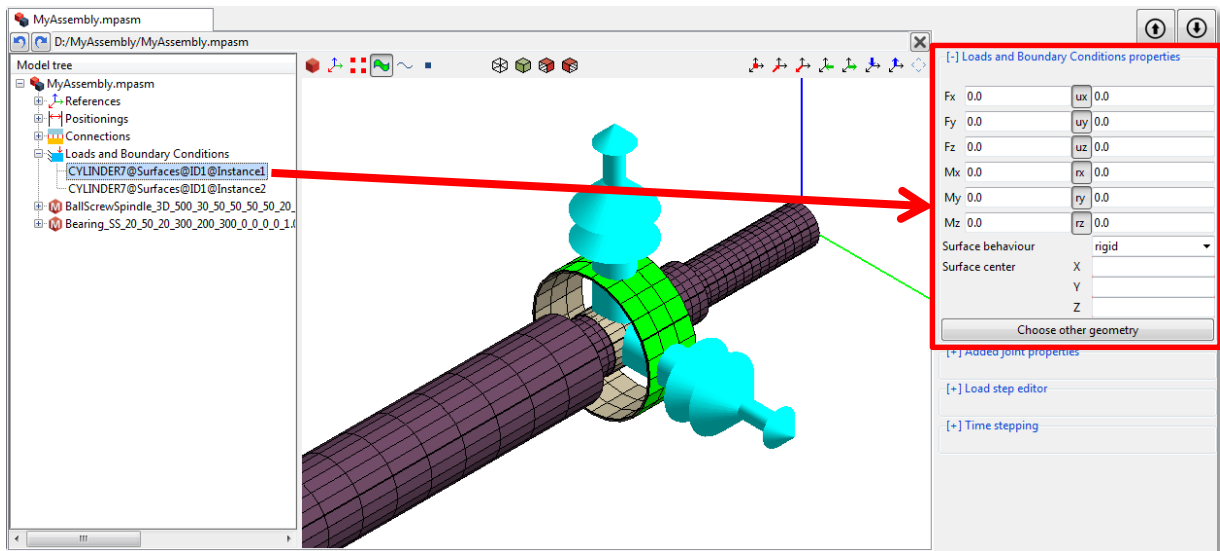
If you are applying a linear velocity to an Origin item, the velocity is considered as an initial condition, meaning it should only be applied in the first load step.

You can apply an acceleration or velocity to subparts of an assembly by selecting the related Origin items. If you select the origin item of the main assembly, then accelerations and velocities are acting on all subparts of the assembly.

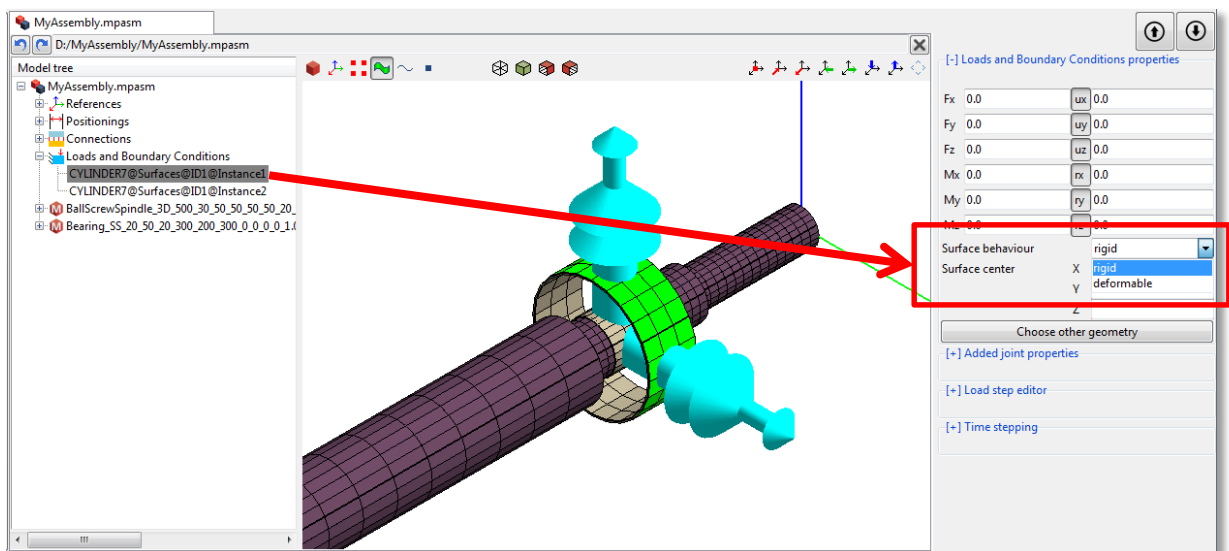
In order to apply LBCs, select the geometry first and then click on one of the buttons “New load” or “New BC”:



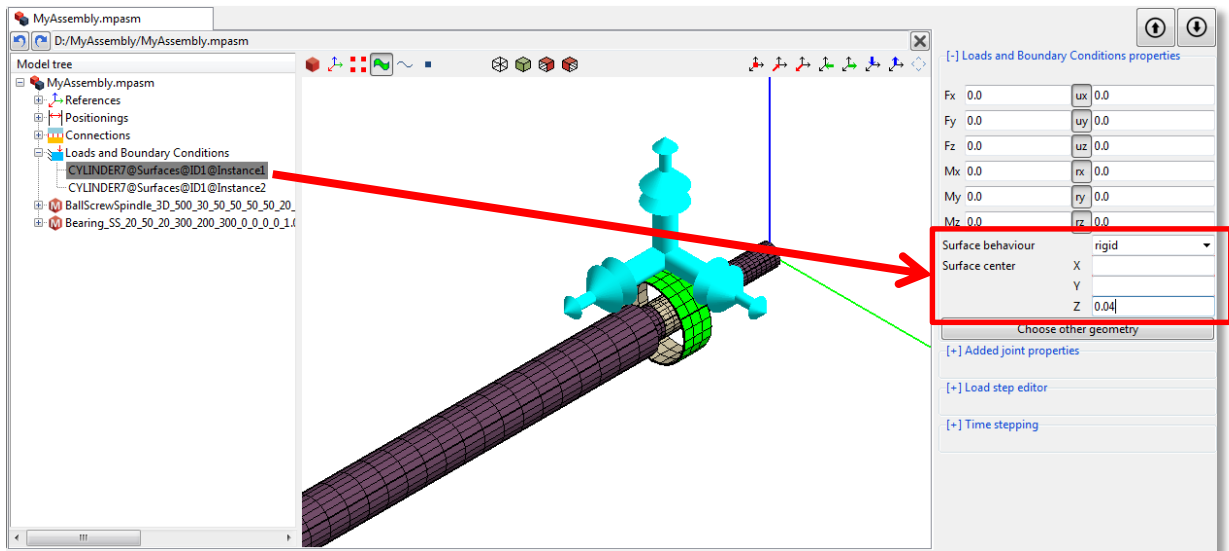
In the “Loads and Boundary Conditions properties” frame you can then specify the **amount of load or displacement** for each individual degree of freedom. These values have only effect in the first time step of a simulation.



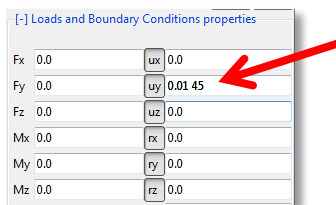
When applying only loads to a surface, the surface behavior can be **flexible or rigid**. You can change this behavior from the "Loads and Boundary Conditions properties" frame:



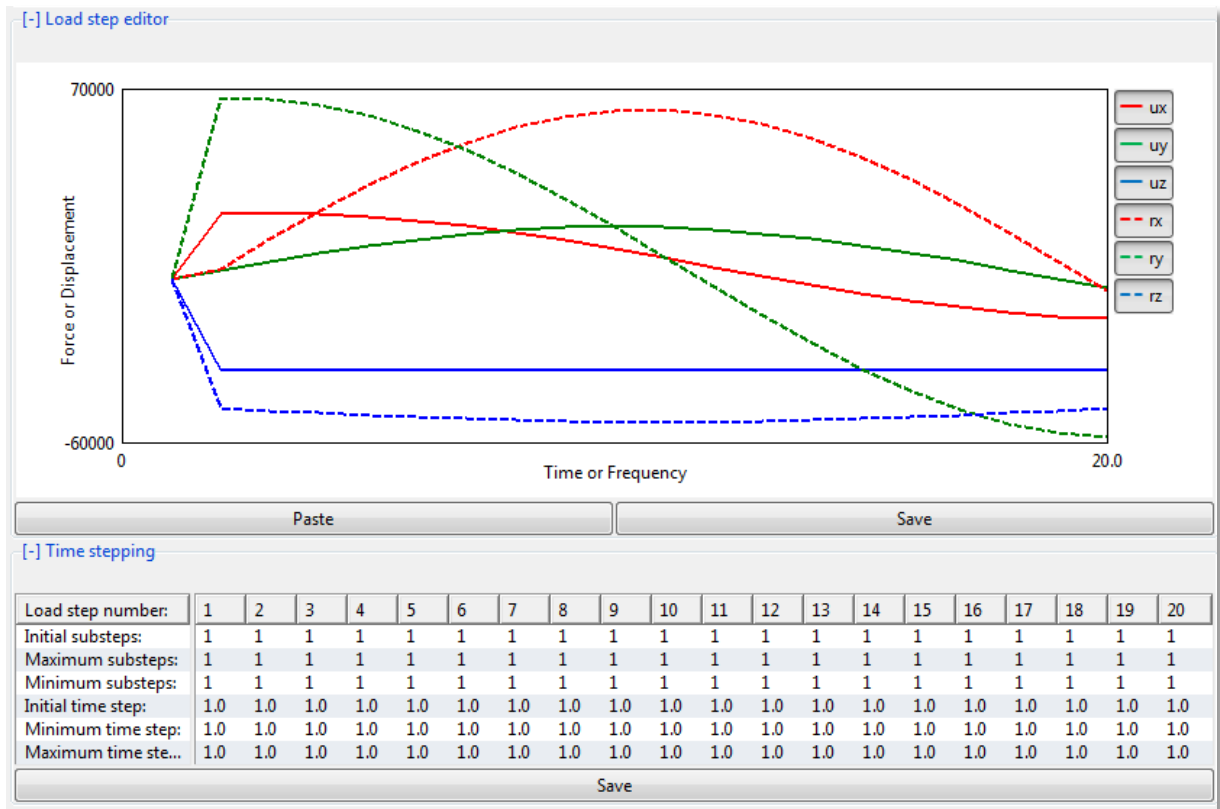
For surface loads and boundary conditions, a pilot node is automatically created at the center of the surface bounding box. For more control over the **location of the pilot node**, you can input your own coordinates relative to the global coordinate system:



In some cases (harmonic analysis), you need to define a **phase angle** for a LBC. In MESHPARTS you simply input both amplitude and phase (in degrees) as numbers separated by one white space into the fields of each degree of freedom:



You can specify LBCs for more than one time step and the number of sub-steps **for each time step** using the "Load step editor" and "Time stepping" frames:



In the "Load step editor" frame, you can select a load or boundary condition curve and **add a new point** by double-clicking on a straight segment of the curve. You can **delete a point** by a right click on that point.

Furthermore, you can **shape the curves** by dragging points or straight segments horizontally or vertically. By holding down the shift key when dragging, the drag resolution increases. You can double-click in the top-left corner of the editor and **enter the exact x/y coordinates** of the point.

You can change the **limits of the axes** (time/frequency/amplitude) by a double click on them. By changing the upper time or frequency limit, all time points will scale accordingly so that the LBC curves will fit the new time interval.

Lastly, you can **import tabular data** from an Excel-Table, text file or similar format. For example, in Excel you select the table area of interest and press CTRL+C to copy it. Then select one or more LBC items and click on the "Paste" button in the Load step editor frame. Click on the "Save" button in the Load step editor frame. The pasted tabular data should have one column for the time and six columns (in general six DOF) for each selected LBC item. So for example if you have two LBC items you will need a table with $1+2 \times 6 = 13$ columns.

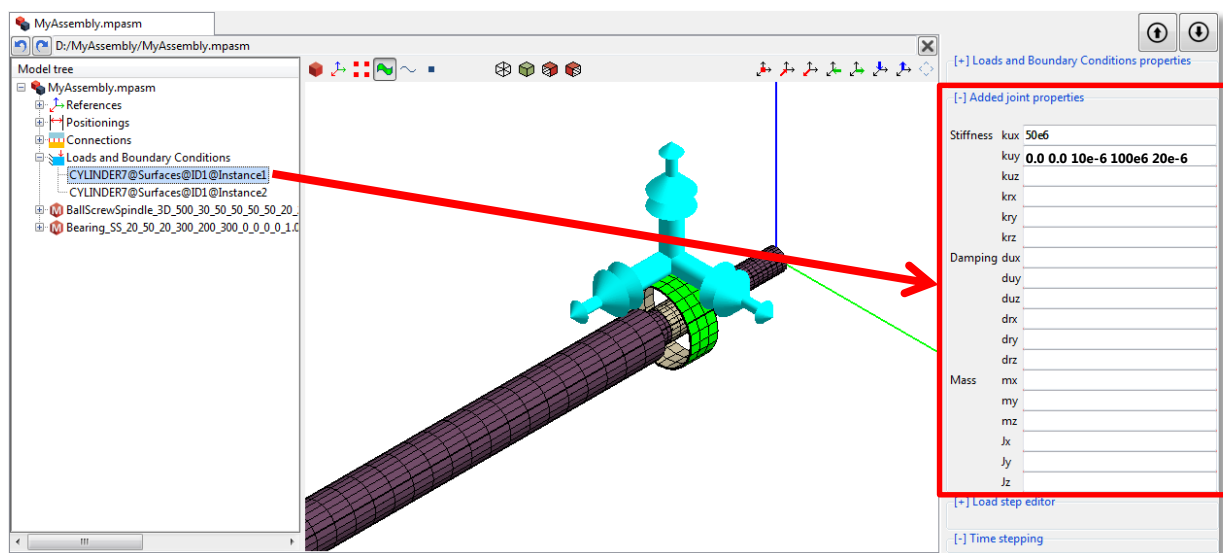
12 Adding discrete joints to an assembly

Typically, discrete joints are formed by two nodes that span a discrete stiffness and/or damping element. Many model components from the Meshparts library already contain discrete joints. These are needed to efficiently model the behavior of linear guides, bearings, ball screw nuts etc.

You can also add discrete joints to FE assemblies rather than to model components. This method is recommended for special FE assemblies that cannot be easily or do not need to be standardized.

Discrete joints are available in the properties frame of defined **connections** or **LBCs**. For connections, discrete joints can only be defined if the connection contains surfaces or points. For LBCs, joints can be added to any type of geometry (surfaces, curves and points).

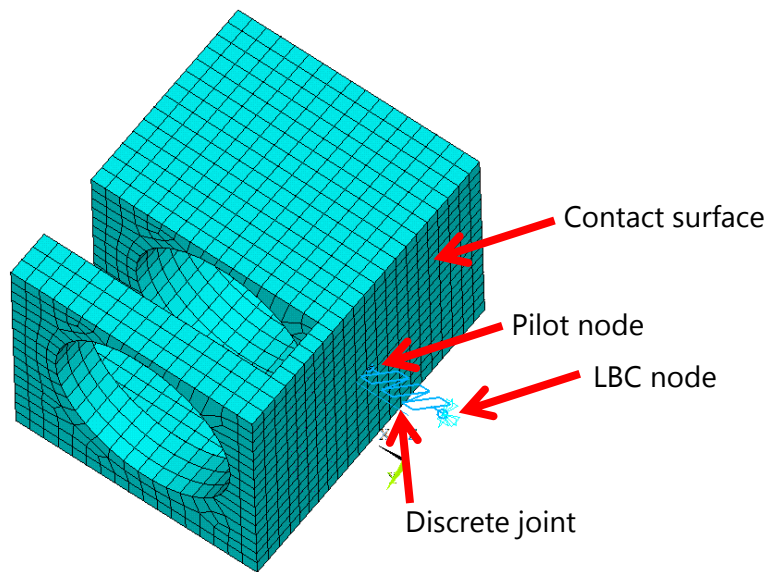
In the following example joint stiffness is added to the outer ring surface of a bearing by selecting the corresponding LBC item in the model tree.



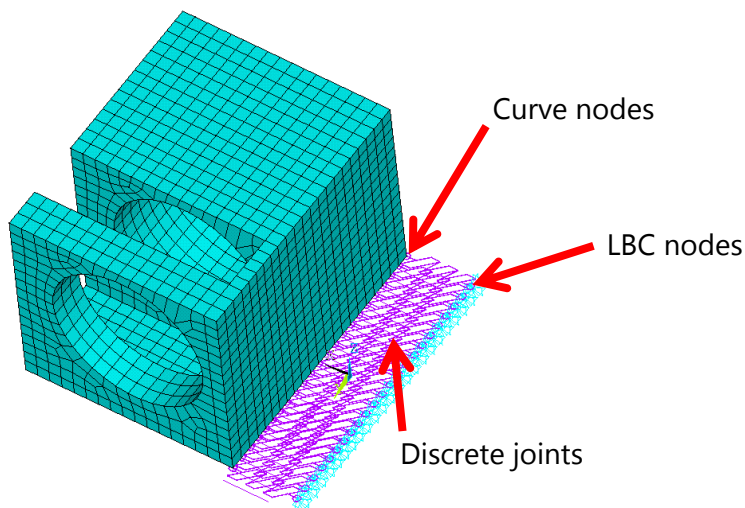
Joint stiffness and damping can be **linear** or **nonlinear**. You can define a joint with linear stiffness or damping by a single value (the stiffness or damping coefficient). Nonlinear joint behavior you can define by inputting displacement-force (stiffness) and velocity-force (damping) pairs, as in the above example for the joint stiffness in the Y degree of freedom. Make sure that the value pairs always contain the origin point (0 0).

In order to add a discrete joint to a LBC, Meshparts internally defines a second node on the same location as the LBC node and attach the discrete stiffness, damping and mass elements. The load or boundary condition is then actually applied to the second node, not to the LBC node (as in the case without added discrete joint).

In the case of a LBC applied to a surface, the added discrete joint is inserted between the LBC node and the pilot node of the contact surface:



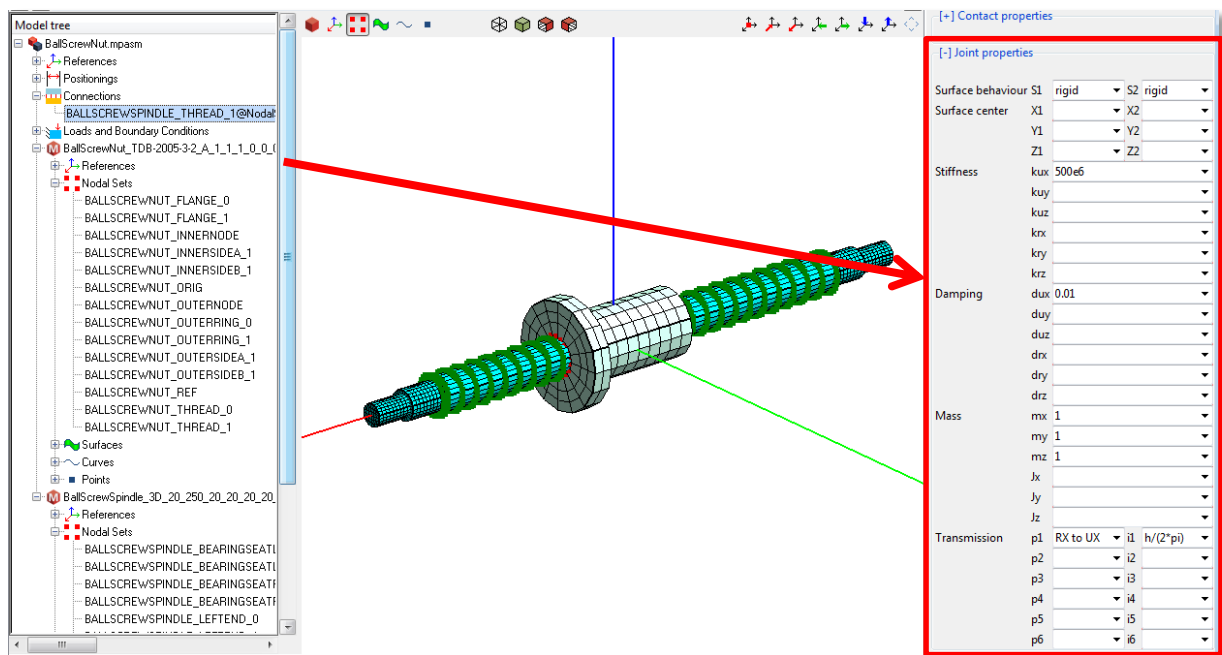
In the case of a LBC applied to a curve, multiple discrete joints are inserted between each LBC node and each curve node. The defined stiffness, damping, mass are equally distributed to each joint:



If you add a discrete joint through a connection item, you can not only define mass, stiffness and damping, but also a **transmission factor** for up to six degrees of freedom combinations (translations and rotations).

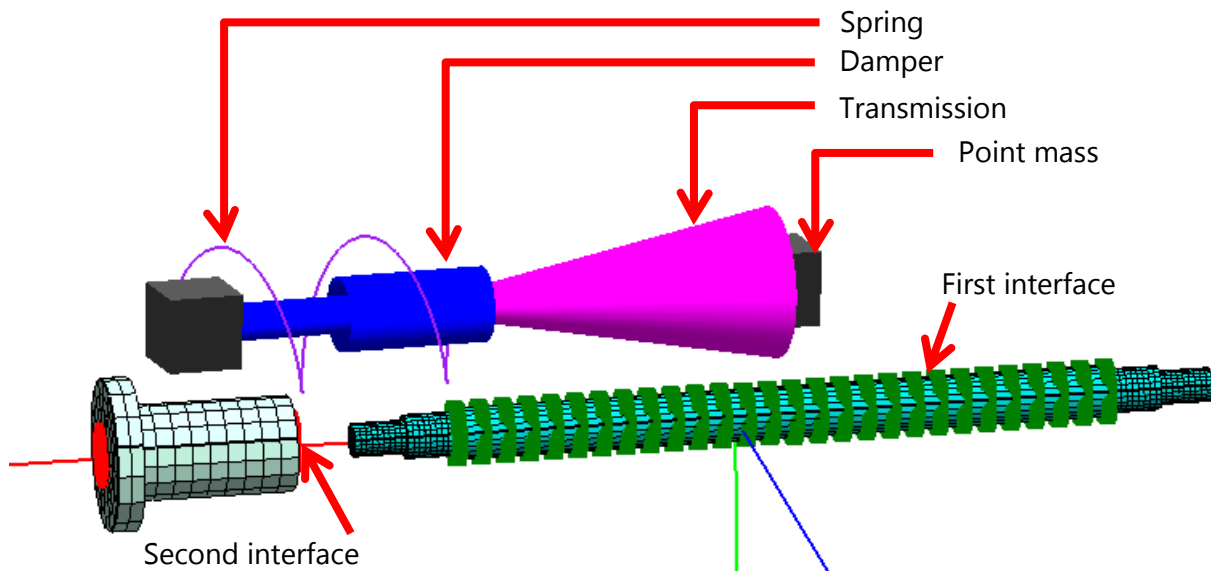
Pay attention to the fact, that the degrees of freedom of the joint are defined in the coordinate system of the assembly that directly contains the joint. If the assembly containing the joint is inserted in another assembly with different orientation, then the joint properties will also adapt to the new orientation.

In the following example, a screw like behavior is defined between the surfaces of two parts along with a linear discrete stiffness and mass equally distributed to the two surfaces:



The screw like behavior is defined in the first degrees of freedom pair as "RX to UX", meaning that the relative translation in X is related to the relative rotation about X by a factor that is input as a parametric expression $\frac{h}{2\pi}$, where h is the thread slope of the screw.

For joints with added transmission, the order of the joint interfaces as it appears in the relation name is very important: first interface, transmission, Spring/Damper, second interface (see below an exploded view of an assembly with added joint).



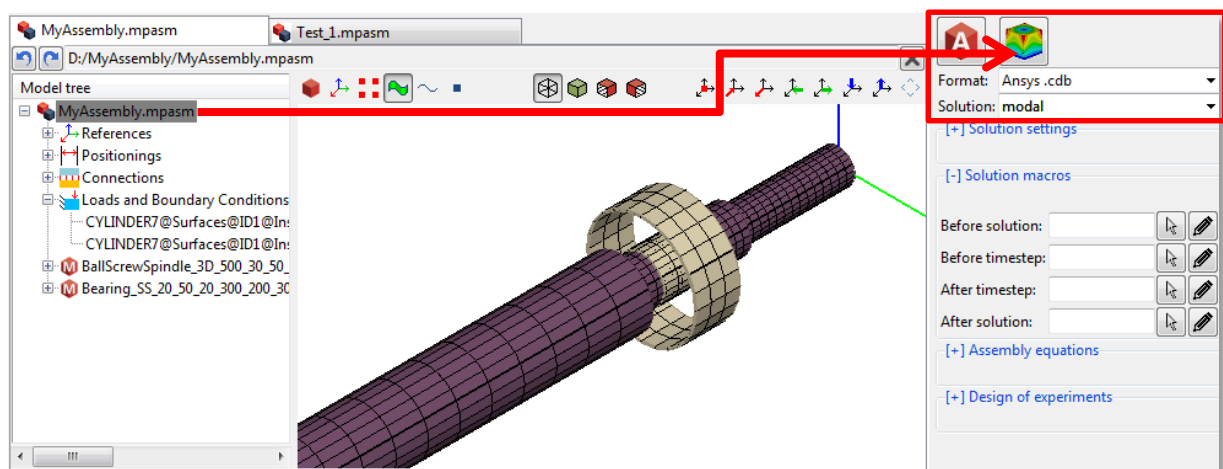
12.1 Nonlinear stiffness

If the nonlinear stiffness of a joint contains only the tensile curve (positive values), the compressive curve (negative values) will be automatically computed by reflecting the tensile curve.

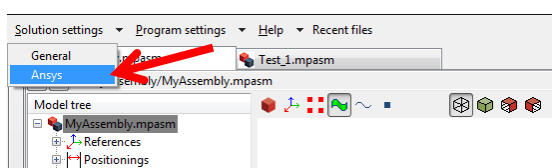
If the nonlinear stiffness of a joint should offer no resistance to compressive loading, then the compressive of the curve should be formed by just one displacement-force pair (typically -1.0 0.0). Thus, a nonlinear spring that offers no resistance to compressive loading could be input in this way: -1.0 0.0 0.0 0.0 10e-6 100e6 20e-6 250e6 30e-6 400e6

13 Solving a FE assembly

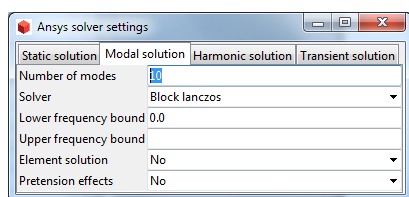
After you define positioning and connection relations and apply loads and/or boundary conditions to an FE assembly, you can obtain a solution of the current FE assembly by selecting the assembly item in the model tree. Select a type of analysis (static, modal harmonic or transient) and click on the solve button:



If you want to setup **more options** select one of the supported solvers (currently Ansys) from the "Solution settings" in the main menu:



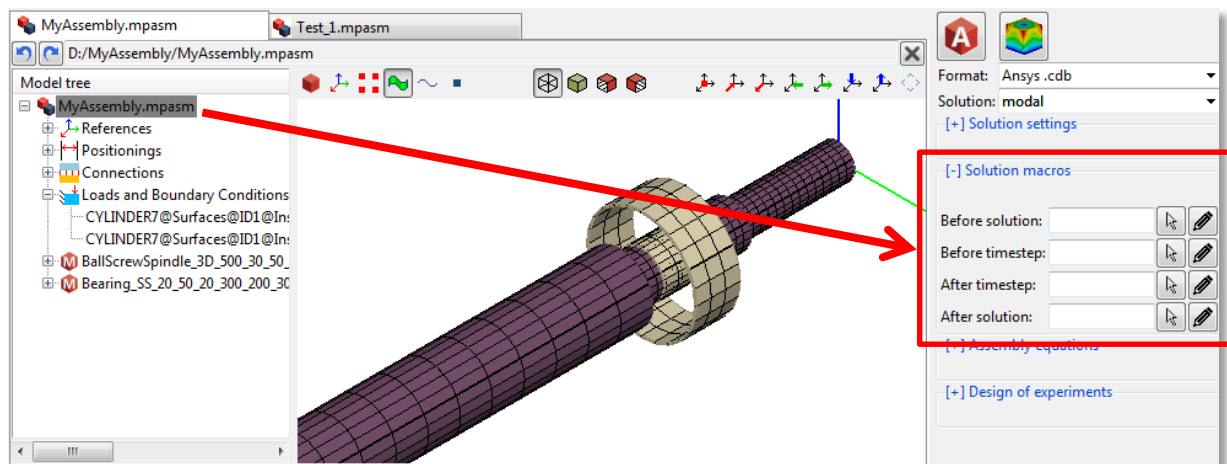
In the new window select the type of analysis you want to set-up (static, modal, harmonic or transient) and adapt the available parameters:



A more advanced feature is the specification of (pre-/post-) **solution macros**. This way you can perform special operations on your model using the integrated scripting language of the

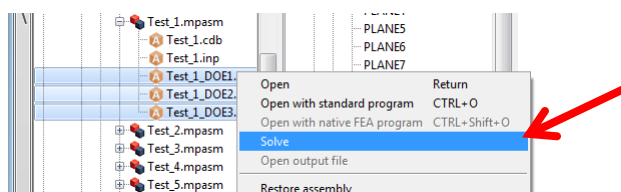
third party solver (e.g. APDL – Ansys Parametric Design Language). You can setup four different macro paths in the “Assembly macros” frame:

- A macro that will be executed **before** the assembly is solved.
- A macro that will be executed **before each time step** of the assembly solution.
- A macro that will be executed **after each time step** of the assembly solution.
- A macro that will be executed **after** the assembly is solved.



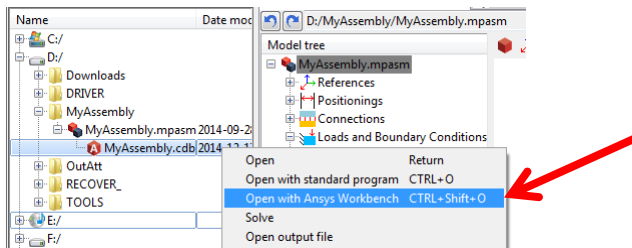
Important! For Ansys models (assembled cdb models), all assembly macros are executed during the solution phase. If you are performing pre-processing operations in one of the solution macros, you have to explicitly enter the pre-processor at the beginning of the macro (/prep7 command) and explicitly enter the solution at the end of the macro (/solu command).

You can solve one or more assembled models also **directly from the explorer tree**. Select one or more assembled model files and choose “Solve” from the context menu. This second method of solving assembled models does not require the corresponding assembly to be open. It also have the advantage over the first method that you can schedule multiple simulations with just one click. The models will then be solved sequentially, one after one:



14 Post-processing the results

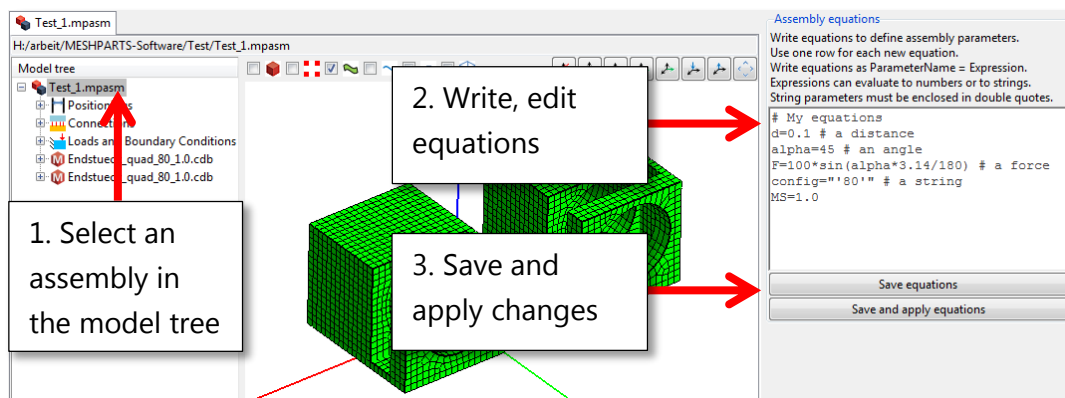
At the current software current release, MESHPARTS does not provide integrated post-processing (evaluating displacements, stresses etc.) of the FE results. Instead, you can right click an exported model file in the explorer tree and choose “Open with Ansys Workbench”, “Open with Ansys MAPDL” or “Open with Abaqus CAE”. You can then perform the post-processing using the native FE program.



15 Working with assembly equations

In many situations, there exists a relation between the parameters of different models or distances of some positioning constraints. In such cases, you can simplify your workflow and accelerate model changes by defining equations that describe the relations between different model parameters of an assembly.

To do so, select an assembly item in the model tree and write new equations or edit available ones in the “Assembly equations” frame:



The syntax of an equation is:

ParameterName = Expression

The parameter names are case sensitive. Do not use empty spaces or special characters in parameter names.

The expression on the right side of the equal sign of an equation can describe:

- a mathematical expression
- a string

By clicking on the “**Save and apply equations**” button, the assembly equations are evaluated and entities linked to these parameters are updated accordingly.

15.1 Using mathematical expressions

In mathematical expressions you can use:

- numbers
- other predefined parameters
- operators (+, -, *, /, **, %)
- parentheses
- functions (abs, sin, cos, tan, asin, etc.)

Numbers can be integers or real numbers. For real numbers use "," for decimal points.

A detailed description of the allowed operators and functions in mathematical expressions is given in the following tables.

Operator name	Symbol	Example	Remark
Addition	+	5 + 3	
Subtraction	-	5 - 3	
Multiplication	*	5 * 3	
Division	/	5.0 / 3.0	If the denominator is an integer, the result will be rounded.
Exponentiation	**	1.5**2	
Modulo	%	5%3	Use only integers with the operator modulo.

Function	What it computes	Example	Remark
abs	The absolute value	abs(-1)	
acos	The arc cosine	acos(1)	The argument of acos must be in the range [-1, 1]. It returns an angle in radians.
asin	The arc sine	asin(1)	The argument of acos must be in the range [-1, 1]. It returns an angle in radians.
atan	The arc tangent	atan(10)	It returns an angle in radians.
atan2	The arc tangent	atan2(1,10)	The arguments of atan2 must be different from 0.
ceil	The smallest integer value not less than the argument	ceil(5.3)	
cos	The cosine	cos(3.14)	The argument of cos must be an angle in radians.
cosh	The hyperbolic cosine	cosh(10)	
double	Converts integer to floating point number	double(1)	
entier	The integer part of any number	entier(5.3)	
exp	The exponential	exp(1)	The basis is the Euler number e.
floor	The largest integer not greater than the argument	floor(5.3)	

fmod	The floating-point remainder	fmod(5,3)	Similar to the % operator. The result is not an integer but a floating point number.
hypot	The length of the hypotenuse of a right-angled triangle	hypot(5,3)	The arguments are the lengths of the two catheti.
int	The integer part of a number	int(5.3)	
isqrt	The integer part of the square root	isqrt(10)	The argument of isqrt must be positive.
log	The natural logarithm	log(10)	The basis is the Euler number e.
log10	The base 10 logarithm	log10(10)	
max	The maximum of one or more numbers	max(5,3,1)	The number of arguments is unlimited.
min	The minimum of one or more numbers	min(5,3,1)	The number of arguments is unlimited.
pow	The first argument raised to the power of the second argument	pow(5,3)	
round	Converts a number to nearest integer	round(5.3)	
sin	The sine	sin(3.14)	The argument of sin must be an angle in radians.
sinh	The hyperbolic sine	sinh(10)	
sqrt	The square root	sqrt(10)	The argument of sqrt must be positive.
tan	The tangent	tan(3.14)	The argument of tan must be an angle in radians.
tanh	The hyperbolic tangent	tanh(10)	
wide	The integer part of a number	wide(5.3)	

15.2 Using strings in equations

Strings in assembly equations must be input between double quotes:

```
ParameterName = "SomeString"
```

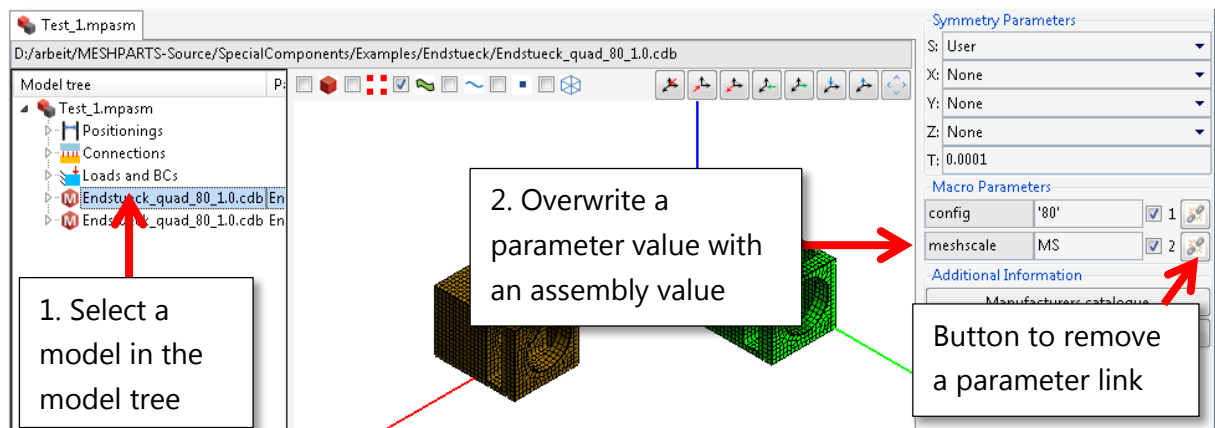
If the equation defines an APDL parameter (which must be input between single quotes) then the parameter is written as follows:

```
ParameterName = 'SomeString'
```

15.3 Linking model parameters

After you define assembly parameters through assembly equations, you can link model parameters to assembly parameters. To do so, you select a model file of an assembly in the

model tree. If you have generated the model file from a macro file, then you can review and change the model parameters in the model parameters frame:

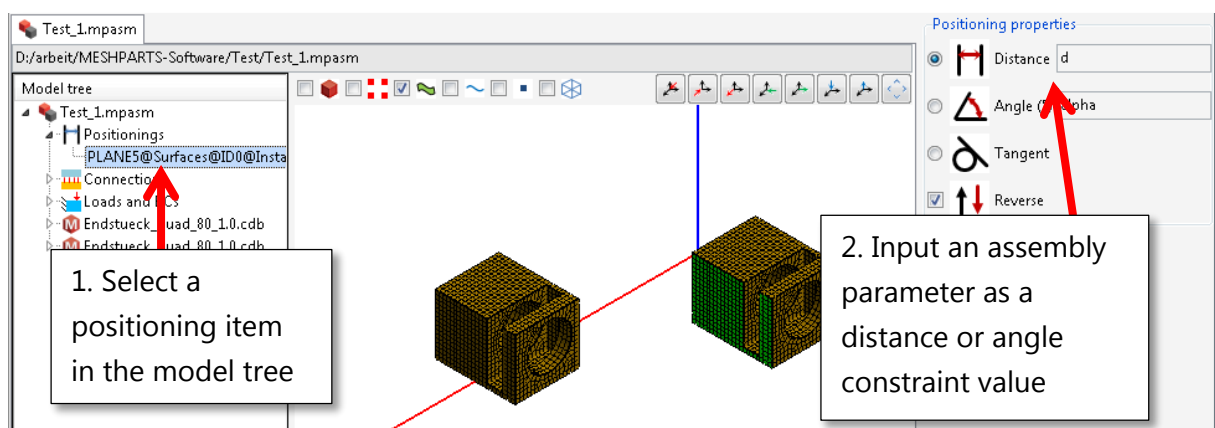


In order to link a model parameter to a predefined assembly parameter, you simply replace the value of the model parameter with the parameter name of the assembly.

In order to remove a model parameter link, click on the button to the right of the parameter value in the model parameters frame.

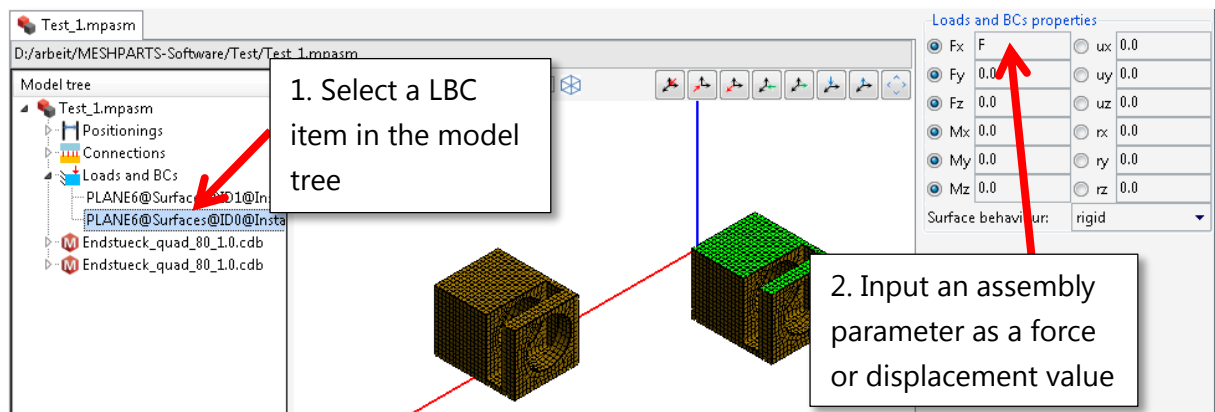
15.4 Linking positioning constraint values

Similar to linking model parameters you can link values of positioning constraints (e.g. distances or angles) to a predefined assembly parameter:

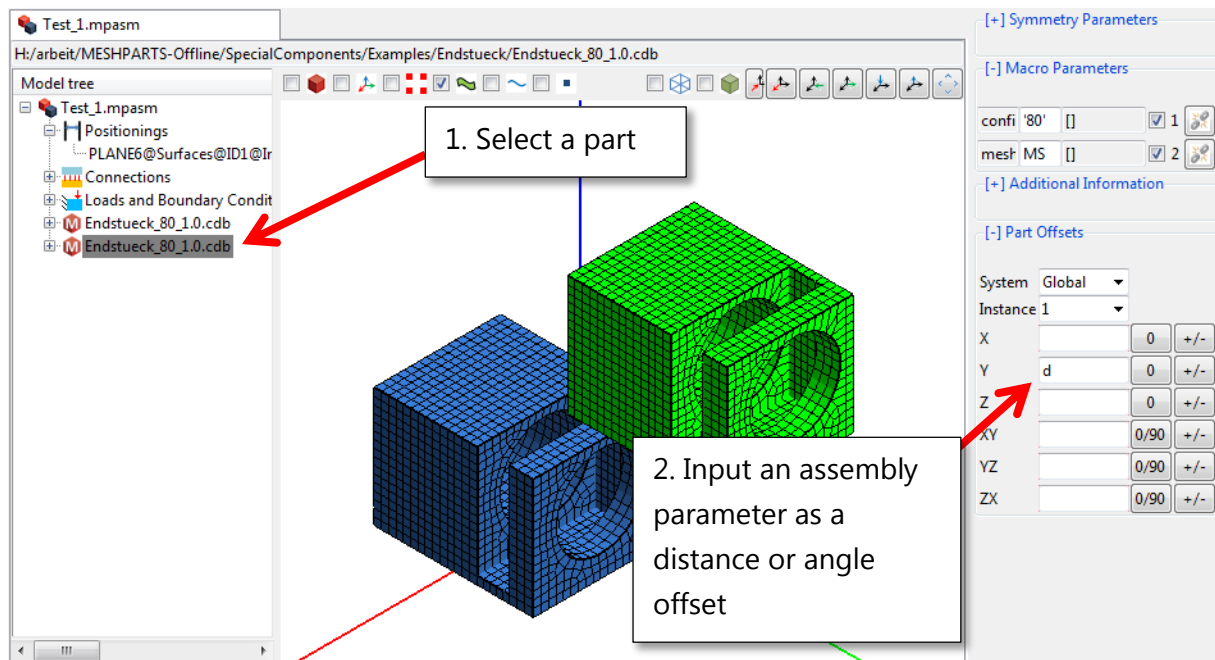


You can remove a parameter link in a positioning constraint by replacing the assembly parameter with a number in the "Positioning properties" frame.

15.5 Linking loads and boundary conditions



15.6 Linking part offsets



15.7 Updating an assembly

Changes to the assembly equations are saved and applied to the assembly by clicking on the **"Save and apply equations"** button of the "Assembly equations" frame. The assembly equations are evaluated and entities linked to these parameters are updated accordingly.

15.8 Import parameters from other assemblies

If you have a main assembly (assembly_A.mpasm) containing other assemblies (assembly_B.mpasm and assembly_C.mpasm) you can make the main assembly parameters visible to the subassemblies simply by adding a line with the path of the main assembly to the "Assembly equations" frame of the subassemblies.

The path to the main assembly can be

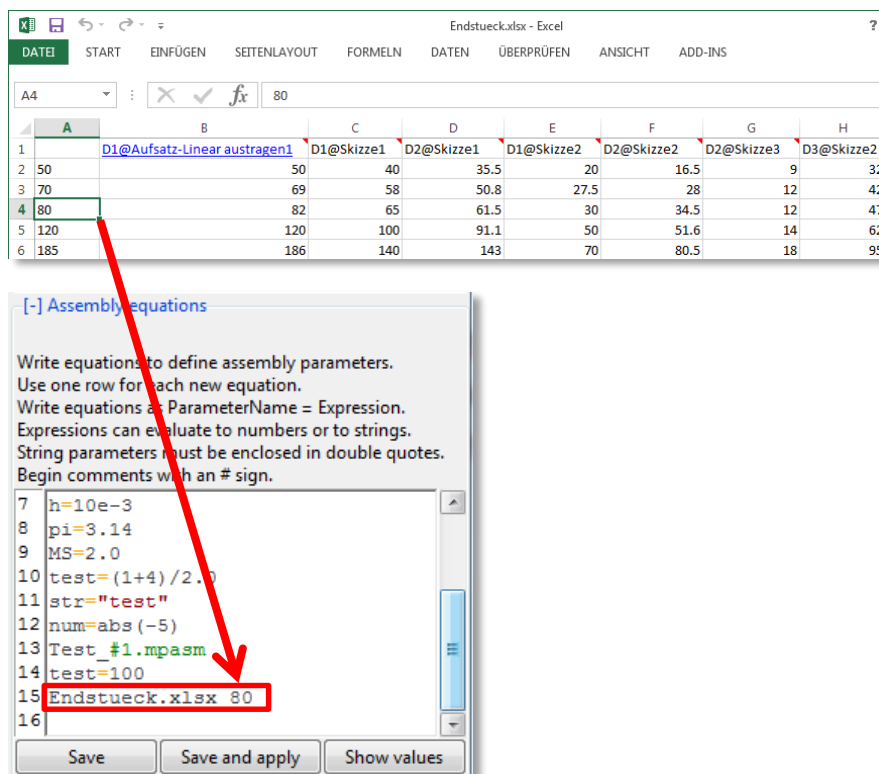
- absolute (e.g. D:/MyMeshpartsModels/assembly_A.mpasm) or

- relative to the path where the subassembly resides (assembly_A.mpasm if both assemblies are in the same folder, ../assembly_A.mpasm if the main assembly is in the parent folder etc.)

Please note the used path separator, which should be a slash, not a backslash.

15.9 Import parameters from an Excel® file

If you have an Excel® file containing different parameter configurations, you can import one of the parameter configurations from Excel® into an assembly simply by adding a line with the path of the Excel® file to the "Assembly equations" frame. After the file path, specify the name of the configuration:



The path to the Excel® file can be

- absolute (e.g. D:/MyMeshpartsModels/mytable.xlsx) or
- relative to the path where the assembly resides (mytable.xlsx if the assembly is in the same folder, ../mytable.xlsx if the assembly is in the parent folder etc.)

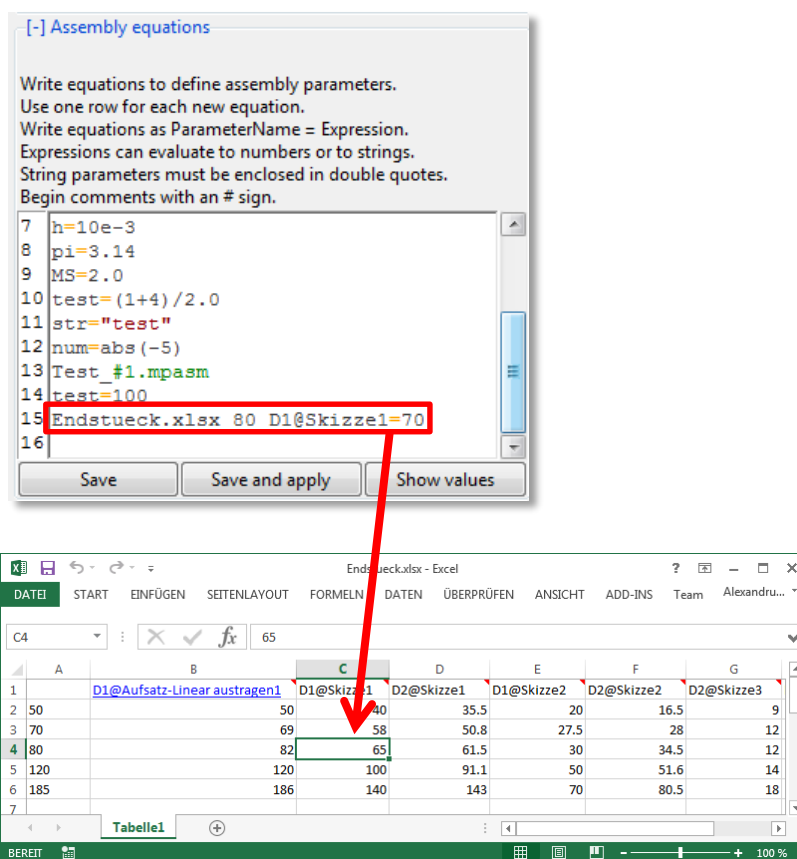
Please note the path separator, which should be a slash, not a backslash.

The Excel® configuration name can be a string (use double quotes), a parameter name or an expression.

15.10 Writing parameters to an Excel® file

Writing parameters to an Excel® configuration file is needed, when some of the assembly models are parameterized through Excel® tables and you want to control those tables from the assembly.

The method to write parameters to an Excel® configuration file is similar to the method of reading/importing parameters from an Excel® configuration file, see chapter 15.10. The only difference is that you have to add the names of the parameters and their values separated by equal signs, see figure below.



16 Replacing a model in an assembly

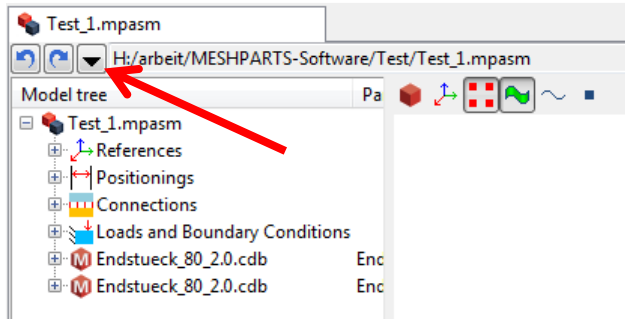
MESHPARTS comes with two ways of replacing a model or assembly in an assembly:

1. Select one or more models or assemblies in the model tree that you want to replace. While holding down the **Shift key** on your keyboard, **drag and drop** another model or assembly from the explorer tree over the selected items in the model tree.
2. Select a parametrical model in the model tree, **change the parameters** of the model and click "Generate new model".

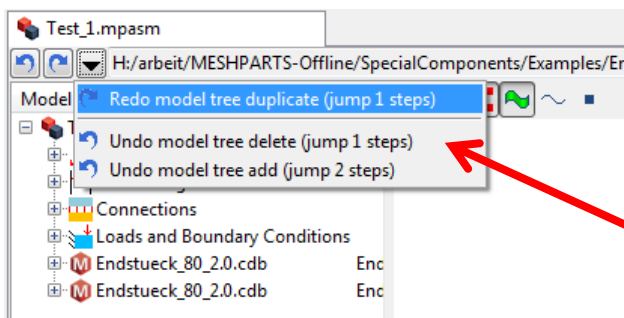
If the new model is similar to the old model, already defined relations (positioning constraints and connections) are maintained.

17 Undoing and redoing changes to an assembly

You can undo or redo changes made to an FE assembly in MESHPARTS by using the three buttons on the top-left corner of the assembly window:



If the last action implied repeated saving of the assembly (e.g. when assembly equations are applied, see Chapter 15), then you would have to press the undo button multiple times, until the needed assembly state is completely restored. In order to simplify this process, there is a third button with an arrow pointing downwards, that you can use in order to jump over multiple undo/redo steps:



18 Generating new model files (parts)

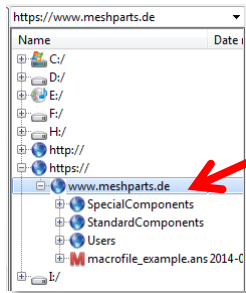
Finite element assemblies consist of one or more model files (parts) or other assembly files.

There are three ways to create (generate) new model files:

- Using the MESHPARTS online library
- Using a finite element software such as Ansys® Mechanical APDL or Ansys® Workbench
- Using macro files

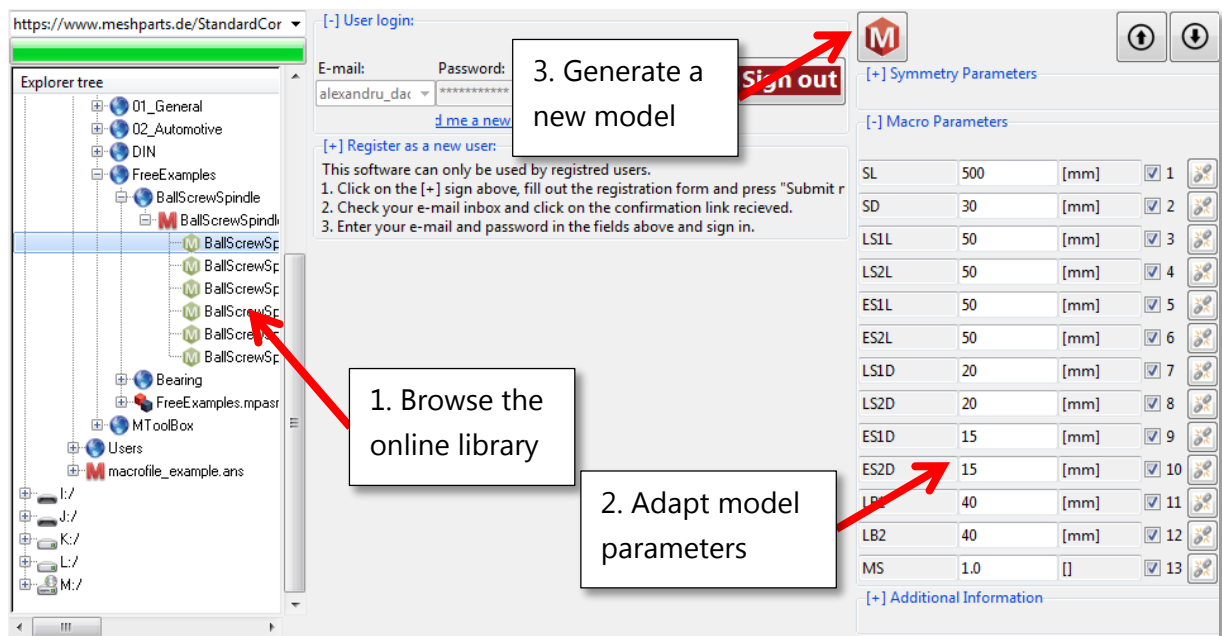
18.1 Using the Meshparts online library

Many standard and manufacturers specific model files you can already find in the MESHPARTS online library. You can access the MESHPARTS online library directly from the explorer tree in the graphical user interface of the MESHPARTS software:



In the MESHPARTS online library, downloading new models is free for all registered users. Generating new models in the cloud is free but requires a valid software license.

You can browse in the explorer tree to a macro or model file of your wish, adapt the model parameters such that they fit to your needs and click on the button “Generate a new model”:



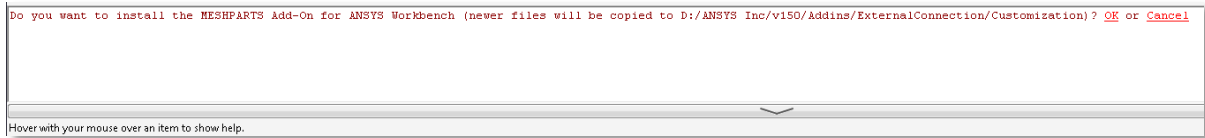
After model generation, the new model will appear as a new child element of the macro file in the explorer tree. In the context menu for this model, you can choose “Download” in order to download the model to your local library path.

18.2 Using a finite element software

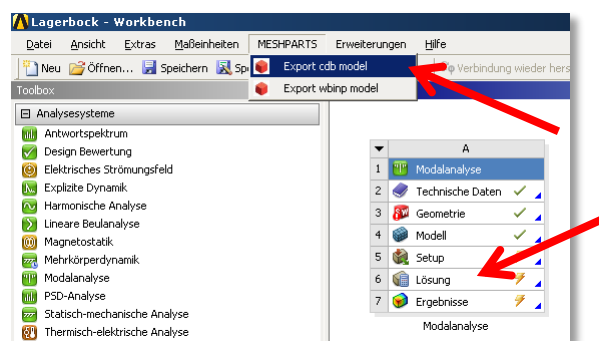
If you cannot find a specific model in the MESHPARTS online library, you can create other model files using a third party finite element software. Currently Ansys® CDB file format is supported by MESHPARTS and many other finite element programs can convert their native file format into Ansys® CDB format.

If you are using Ansys® Workbench to generate your finite element models, MESHPARTS provides a small add-on that simplifies the export of Ansys® Workbench models into the native CDB file format.

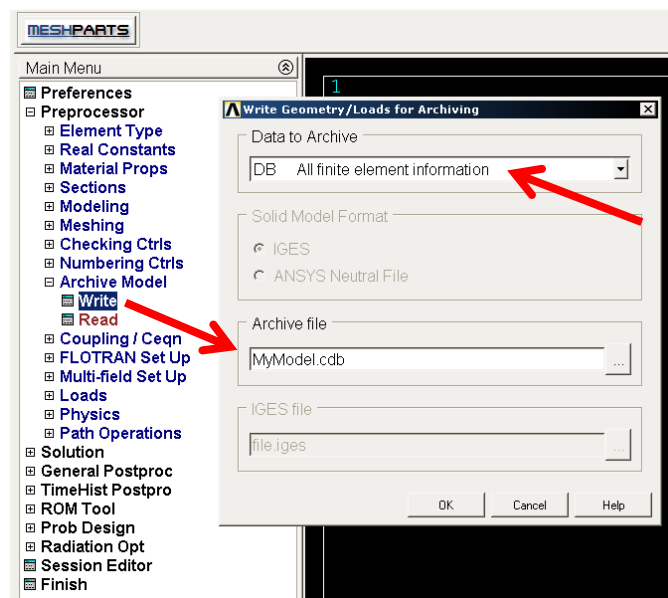
You can quickly install the MESHPARTS add-on for Ansys® Workbench when you start the MESHPARTS software. If the add-on is not already available, you will see a message asking you if you wish to install the add-on:



After installation, a new menu item will be visible in the Ansys® Design Space menu bar. Choose Main Menu, MESHPARTS, Export cdb model. The Ansys® Workbench project must contain a system with a solution container as marked in the picture below.



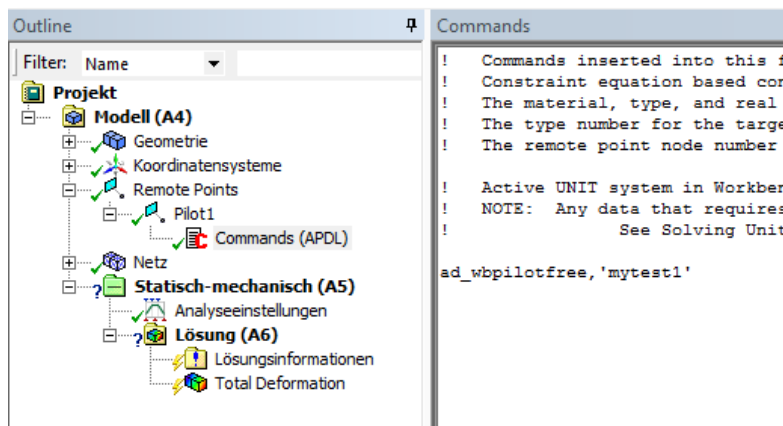
If you are using Ansys® Mechanical APDL choose Main Menu, Preprocessor, Archive Model, Write. In the new dialog window, choose "DB All finite element information" and a model file name.



You can assign names to special interfaces in models created with both Ansys® MAPDL or Ansys® Workbench. The method in Ansys® MAPDL can be found under "nodal sets" in the Ansys® Manual.

In Ansys® Workbench the method is different whether the interface is a geometry item (surface, curve or point) or a free point (Meshparts pilot nodes). In the first case you simply define a named selection. In the second case you define an external point and then attach a command object to it and insert the name of a special Meshparts APDL macro (ad_wbpilotfree) along with then name of the interface of that pilot node:

ad_wbpilotfree,'<name_of_the_interface>'

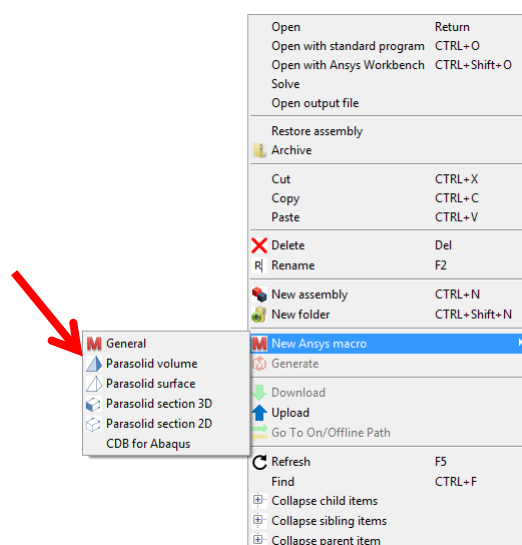


18.3 Using macro files

This is the most powerful method for creating new model files as it is completely automated and model parameters can be changed very fast. The same method is used for the MESHPARTS online library.

Currently only Ansys macro files are supported.

Right click a folder in the explorer tree. From the context menu, choose "New Ansys Macro File" and one of the macro types depicted in the picture below.

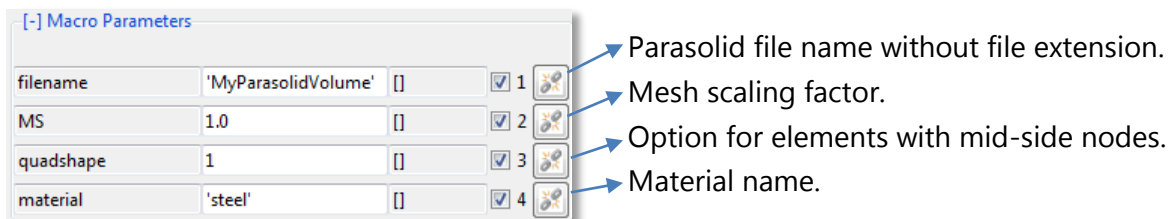


The meaning of the different macro types is:

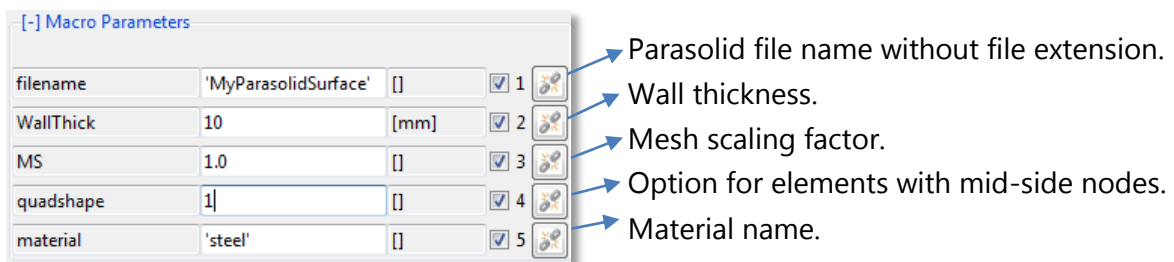
- **General:** A collection of useful, general purpose commands. This macro type requires programming knowledge as it must be adapted to different tasks.
- **Parasolid volume:** Out-of-the-box parametric macro, which imports a Parasolid volume and generates a high quality mesh with assigned material.
- **Parasolid surface:** Out-of-the-box parametric macro, which imports a Parasolid surface and generates a high quality mesh with assigned material and thickness.
- **Parasolid section 3D:** Out-of-the-box parametric macro, which imports a Parasolid plane surface and generates a high quality mesh by extruding the plane surface to a prismatic volume. The area must lie in the XY plane.
- **Parasolid section 2D:** Out-of-the-box parametric macro, which imports a Parasolid plane surface and generates a high quality mesh by extruding the contour of the plane surface to a prismatic surface. The area must lie in the XY plane.

The macro will be defined as a new file in the selected folder. By selecting a macro file, the macro parameters are listed on the right hand side of the GUI. Here is the description of the macro parameters for the macro types:

Parasolid volume



Parasolid surface



Parasolid section 3D

[-] Macro Parameters

filename	'MyParasolidSection'	[]	☒	1	
Length	1500	[mm]	☒	2	
Angle1	0	[°]	☒	3	
Angle2	0	[°]	☒	4	
Angle3	0	[°]	☒	5	
MS	1.0	[]	☒	6	
quadshape	1	[]	☒	7	
material	'steel'	[]	☒	8	

Parasolid file name without file extension.

Length of the extruded volume.

Off-plane rotation angle of first ending area.

Off-plane rotation angle of second ending area.

Rotation angle of second ending area about the middle axis of the extruded volume.

Mesh scaling factor.

Option for elements with mid-side nodes.

Material name.

Parasolid section 2D

[-] Macro Parameters

filename	'MyParasolidSection'	[]	☒	1	
Length	1500	[mm]	☒	2	
Angle1	0	[°]	☒	3	
Angle2	0	[°]	☒	4	
Angle3	0	[°]	☒	5	
MS	1.0	[]	☒	6	
quadshape	1	[]	☒	7	
material	'steel'	[]	☒	8	
WallThick	10	[mm]	☒	9	

Parasolid file name without file extension.

Length of the extruded volume.

Off-plane rotation angle of first ending area.

Off-plane rotation angle of second ending area.

Rotation angle of second ending area about the middle axis of the extruded volume.

Mesh scaling factor.

Option for elements with mid-side nodes.

Material name.

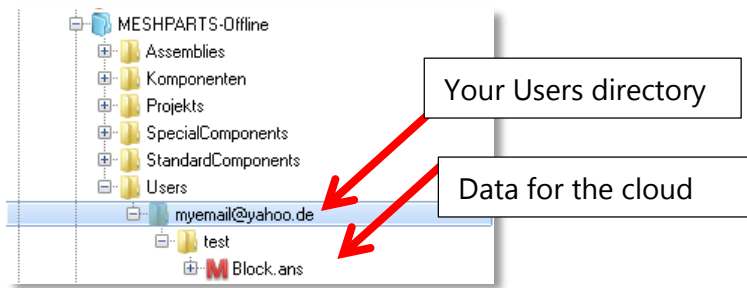
Wall thickness.

You can also edit the content of the macro file using a text editor of your wish. If you choose "Open with standard program" from the context menu, the macro file will be opened with the program that is set to open files with the .ans extension on your system. You can read more about how to use Ansys® macro files under MEHSPARTS in chapter 19.

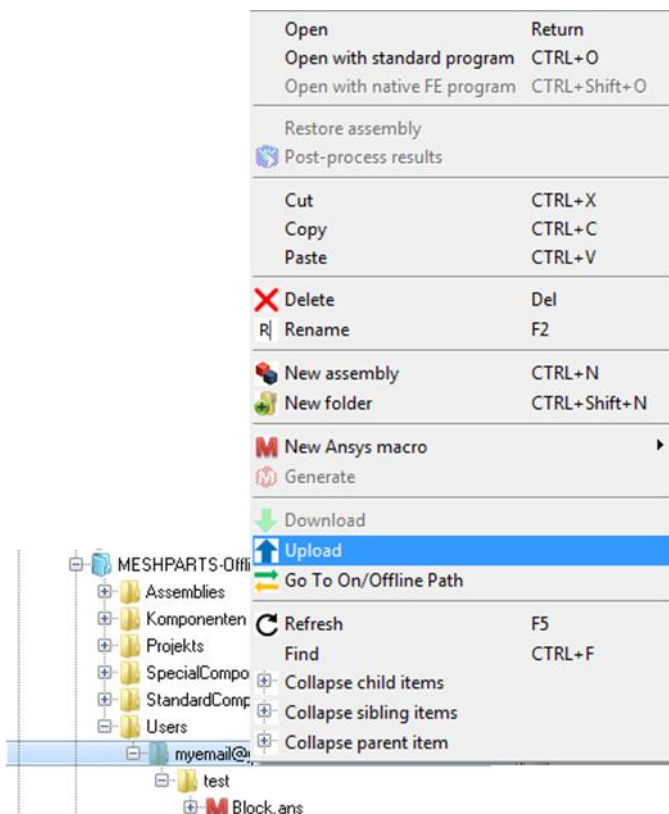
18.4 Uploading macro files to the online library

Meshparts macro files are typically executed by a corresponding third-party FE program (e.g. Ansys). If the meshparts macro file cannot be executed due missing installation of the required third-party program, you can alternatively upload and run the macro files in the MESHPARTS online library.

Place all the macro files and additional files (CAD import files) that you want to run in the cloud in **your Users directory** of the offline library. Your Users directory is actually named after your e-mail address, so it is crucial that you use the same user directory as your e-mail address:



You can then upload the files and/or directories to the Meshparts online library using the context menu:



After uploading the content to the Meshparts online library you can run the macro files as usual but without having the need for the third party FE program to be installed on your own system.

18.5 Model units

You can define new models using any combination of units. However, the MESHPARTS online library contains models defined according to the [International System of Units \(SI\)](#). If you plan to assemble models from the MESHPARTS online library with self-generated models then you should create your models using the same units:

Quantity name	Unit name	Unit symbol
Length	meter	m
Mass	kilogram	kg

Time	second	s
Force	Newton	N

19 Working with Ansys® macro files

A very good starting point with Ansys® macro files are the templates that MESHPARTS provides (see chapter 17)

The inner structure of the macro file is based on the ANSYS® user library structure, which is actually a collection of APDL macros and sub macros:

- MAIN
 - DEFINEGEOMETRY
 - DEFINEPROPERTIES
 - ASSIGNPROPERTIES
 - GENERATEMESH
 - DEFINECOMPONENTS
 - INTNODES

The MAIN macro expects up to 18 arguments (arg1, arg2 ..., arg9, ar10, ar11 ..., ar18). The declaration of arguments in the MAIN macro should be:

```
parametername=argumentname ! comment [units] {type range}
```

As an example, if you would like to specify the value of the parameter `length` then place

```
length=arg1/1000 ! length of block [mm] {integer >0}
```

in the MAIN macro of your macro file. If you use more than 9 arguments, then take into account that you have to use ar10, ar11 ..., ar18 and **not** arg10, arg11 ..., arg18.

The six sub macros listed above provide regularity during the creation of macro files and maintain a high level of flexibility. The **DEFINEGEOMETRY** macro should define ANSYS geometry entities such as keypoints, lines, areas or volumes. Universal CAD files can also be imported here. The **DEFINEPROPERTIES** macro should define element types, material properties, real constants, section properties. The **ASSIGNPROPERTIES** macro is meant for assigning material and mesh size properties to the geometry entities and could contain for example KATT, LATT, AATT or VATT, LESIZE commands. The **GENERATEMESH** macro should define the finite element entities such as nodes and elements and all other related information. The **DEFINECOMPONENTS** macro is intended to define node and element sets (see APDL command CM), which are of special interest during the solution or post processing phase. Finally the **INTNODES** macro defines the nodal sets, which represent the interface nodes to other components.

19.1 Validity of parameters

If you would like to your parameters input to be checked for validity place at the end of your parameter comment a validity expression enclosed by curly braces defining the type and range of the parameter. Typical validity expressions are listed below.

Parameter Type	Validity expression
integer	{integer}
positive integer	{integer >0}
negative integer	{integer <0}
integer between 0 and 2	{integer >=0 <=2}
one of the integers	{integer 5,9,20}
integer range with steps of 1	{integer 1:100}
integer range with steps of 10	{integer 1:10:100}
Real	{real}
positive real	{real >0}
negative real	{real <0}
one of the reals	{real 5,9,10.5}
real range	{real 0.1:2.0}
real range with discrete steps	{real 0.1:0.1:2.0}
alphanumeric string (up to 32 characters enclosed in single quotes)	{string}
one of the strings	{string 'flexible','rigid'}
component name	{component}
parameter relation	{integer >Di <=Da}
parameter relation with equation	{integer >0 <=(Da-Di)/2}

19.2 Parameters configurations

In some cases, the number of parameters needed for creating a model file is higher than the maximum number of APDL macro parameters, which is 18. At the same time, more parameters can be grouped together to parameters configurations, thus simplifying the input of model configurations.

Parameters configurations can be defined in Excel-Tables or CSV tables with special format, so called "feature files" (German: Merkmaldateien, file extension *.TAB).

MESHPARTS can read Excel or TAB files automatically, when a model file is generated. In order to accomplish this, a line of code must be inserted in the MAIN part of the macro file:

```
~eui, meshparts::ReadExcelConfig Coupling.xlsx %config%
```

The Excel-File Coupling.xlsx should be formatted similar to the example in picture below. The first row of the Excel table contains the names and units of parameters. The first column contains the configurations names. One of the configuration names is included in the import command shown above. The configuration name must be enclosed by two percent characters.

The screenshot shows an Excel window titled 'Coupling.xlsx - Microsoft Excel'. The active sheet is 'Tabelle1'. The table has 7 columns (A-G) and 6 rows (1-6). The first row contains parameter names with units in brackets. The second row has values for 'small', and the third and fourth rows have values for 'midle' and 'big' respectively. The value '15' in cell F2 is highlighted with a black border.

	A	B	C	D	E	F	G
1	Config []	JN [kg*mm^2]	JZK [kg*mm^2]	ktstat [Nm/rad]	krstat [N/mm]	L [mm]	
2	small	0.015	0.002	1.78	43	15	
3	midle	0.03	0.006	3.15	82	15	
4	big	0.045	0.01	5.16	154	15	
5							
6							

Optionally the parameters configurations can be defined in the Excel file in different sheets. In that case the sheet number (integer) or name (string) can be provided to the import command:

```
~eui, meshparts::ReadExcelConfig Coupling.xlsx %config% %sheet%
```

Finally the configurations name and optionally the sheet number or name must be declared as an APDL macro parameter before the import command. In our example:

```
config=arg1 ! configuration name [] {string}
```

```
sheet=arg2 ! sheet name [] {string}
```

```
~eui, meshparts::ReadExcelConfig Coupling.xlsx %config% %sheet%
```

19.3 Units in Excel configurations

As described in the previous section, the first row of an Excel configuration file contains the parameters names. If you specify the units of the parameters, Meshparts will convert the parameters values to SI units upon reading the configuration file an generating new models based on it.

You can specify parameters units by writing the units enclosed in rectangular brackets into the cell containing the parameter name.

When specifying units, you can use one of the listed symbols in the next table. You can also use the operators *, / or ^ to combine units (e.g. N/mm^2).

Symbol	Conversion factor to SI units	Full name
s	1	Seconds
ms	1.00E-03	Milliseconds
µs	1.00E-06	Microseconds
us	1.00E-06	Microseconds
ns	1.00E-09	Nanoseconds
min	60	Minutes
h	360	Hours
m	1	Meters

cm	1.00E-02	Centimeters
mm	1.00E-03	Millimeters
μm	1.00E-06	Micrometers
um	1.00E-06	Micrometers
nm	1.00E-09	Nanometers
km	1.00E+03	Kilometers
kg	1	Kilogram
g	1.00E-03	Gram
mg	1.00E-06	Milligram
μg	1.00E-09	Microgram
ug	1.00E-09	Microgram
ng	1.00E-12	Nanogram
t	1.00E+03	Tons
kt	1.00E+06	Kilotons
Mt	1.00E+09	Megatons
Gt	1.00E+12	Gigatons
grad	0.015707963	Grads
Grad	0.017453293	Degrees
°	0.017453293	Degrees
arcmin	0.000290888	Arcminute
arcsec	4.85E-06	Arcsecond
rad	1	Radian
mrاد	1.00E-03	Milliradian
μrad	1.00E-06	Microradian
urad	1.00E-06	Nanoradian
nrad	1.00E-09	Nanoradian
N	1	Kilonewton
kN	1.00E+03	Kilonewton
MN	1.00E+06	Meganewton
GN	1.00E+09	Giganewton
cN	1.00E-02	Centinewton
mN	1.00E-03	Millinewton
μN	1.00E-06	Micronewton
uN	1.00E-06	Micronewton
nN	1.00E-09	Nanonewton
Nm	1	Newtonmeter
kNm	1.00E+03	Kilonewtonmeter
MNm	1.00E+06	Meganewtonmeter
GNm	1.00E+09	Giganewtonmeter
Pa	1	Pascal
kPa	1.00E+03	Kilopascal
MPa	1.00E+06	Megapascal
GPa	1.00E+09	Gigapascal
A	1	Ampere
mA	1.00E-03	Milliampere

V	1	Volt
mV	1.00E-03	Millivolt
Ohm	1	Ohm
mOhm	1.00E-03	Milliohm
H	1	Henry
mH	1.00E-03	Millihenry

19.4 CAD import

The MESHPARTS software provides automatic CAD model update. This feature is very useful when complex geometric entities are first generated with the help of specialized CAD software (such as SolidWorks®), then exported to an universal file format (such as Parasolid) and finally imported into ANSYS Mechanical APDL. These three steps can be performed automatically by the MESHPARTS software. For this purpose following rules must be taken into account:

- The supported CAD files are currently of type SLDPRT (SolidWorks® Parts).
- The supported CAD universal format is Parasolid and ACIS (ending X_T and SAT).
- The CAD files must lie in the same folder as the universal format files.
- The name of the CAD universal format file must begin with the name of the CAD file and end with an optional configuration name, in which case an underscore must be used for separation.

These file naming rules are exemplified in the following.

Suppose you have a macro file Guide_v1.ans, which imports the Parasolid Guide_v1_long.x_t exported from Guide_v1.sldprt. In this case "v1" is the version name and "long" is the configuration name. The CAD file Guide_v1.sldprt should provide at least one configuration with the name "long". In order to import the Parasolid file Guide_v1_long.x_t place the following command line into the sub macro GENERATEGEOMETRY of the macro file Guide_v1.ans:

```
~parain,Guide_v1_long,x_t,,all
```

If more than one macro file share the same CAD file then do not use any version name in the name of the CAD file. For example if Guide_v1.ans and Guide_v2.ans use the same CAD file, then the CAD file name should be Guide.sldprt. Furthermore the CAD file could provide more than one configuration, e.g. "long" and "short". In this case you should specify in the macro file which configuration should be imported, e.g.:

```
~parain,Guide_long,x_t,,all
```

or

```
~parain,Guide_short,x_t,,all
```

The configuration name could also be specified by a parameter name of type string:

```
config='short'
```

```
~parain,Guide_%config%,x_t,,all
```

The **recommended method** is to specify the configuration name through a macro argument with validity checking:

```
config=arg1 ! configuration name [] {string 'long','short'}
```

```
~parain,Guide_%config%,x_t,,all
```

If you do not specify the configuration name, then the last active configuration will be imported:

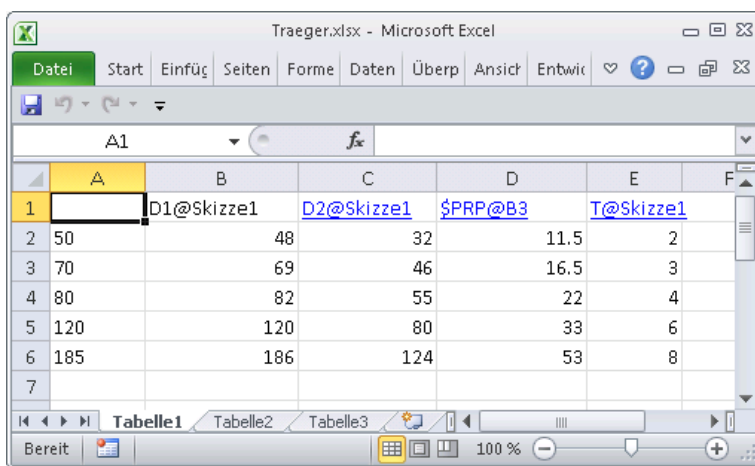
```
~parain,Guide,x_t,,all
```

If the Parasolid file is not available, it will be automatically generated, if the corresponding CAD file is available. If the Parasolid file is available but older than the corresponding CAD file, it will also be automatically updated. This task can only be performed, if the required CAD software (currently only SolidWorks®) is installed on your system. If you are using other CAD software than SolidWorks® please contact us.

The coordinate system used for exporting Parasolid and ACIS files defaults to the origin of the CAD model. If a user defined coordinate system named "Export" is available in the CAD model, then this system will be used instead.

19.5 Sharing parameters configurations between CAD and macro file

When a macro file is using a parametric CAD model it can be advantageous to share parameter configurations. CAD systems such as SolidWorks® or Creo Elements/Pro® (former ProEngineer®) are capable of reading parameters configurations defined in Excel tables. In SolidWorks® the format of the Excel file is similar to the format described in chapter Parameters configurations. The parameters names must match the parameters used in the SolidWorks® model and always contain the @ character.



	A	B	C	D	E	F
1		D1@Skizze1	D2@Skizze1	\$PRP@B3	T@Skizze1	
2	50	48	32	11.5	2	
3	70	69	46	16.5	3	
4	80	82	55	22	4	
5	120	120	80	33	6	
6	185	186	124	53	8	
7						

Furthermore user defined parameters which are not necessary related to geometric properties of the CAD model (e.g. Poisson ratio) can also be included in the Excel file. In this case and according to the SolidWorks® user manual, the parameter name must begin with \$PRP, as shown in the cell D1 of the Excel table above.

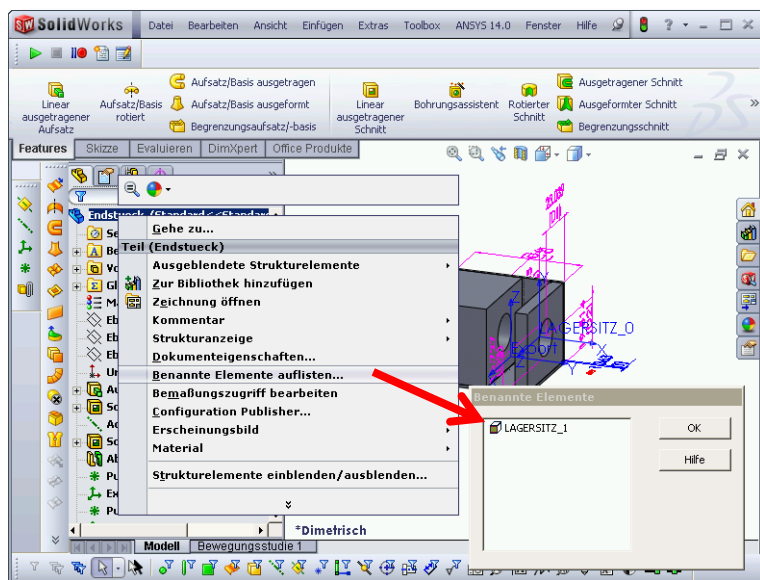
Provided these rules, the same Excel table with parameters configurations can be used in both SolidWorks® and macro files.

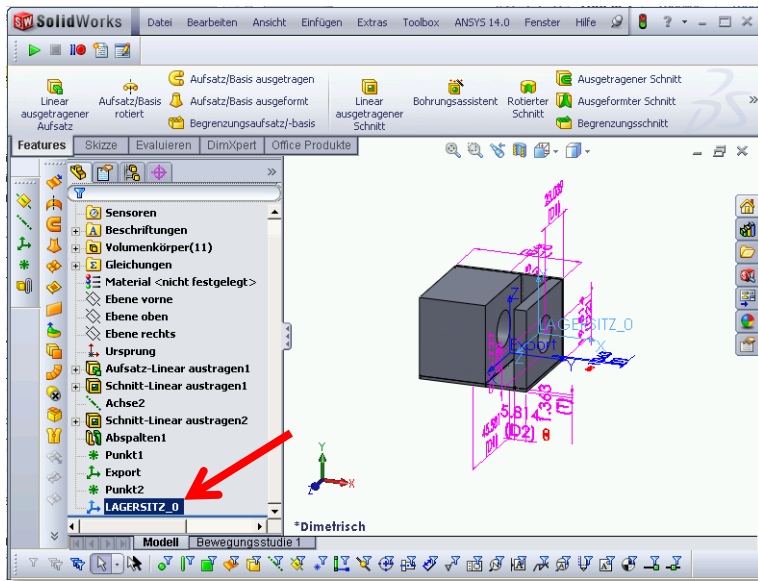
In the macro file, parameters cannot contain special characters and therefore all special characters of the SolidWorks® parameters (e.g. @ and \$) are automatically replaced by underscore characters by the MESHPARTS software. This must be taken into account when these parameters are used in the macro file (e.g. _PRP_B3 must be used instead of \$PRP@B3).

19.6 Interface import from CAD

When assembling model components with MESHPARTS the definition of model interfaces can be very helpful. When CAD models (currently SolidWorks® parts) are imported from within a macro file, the CAD model is checked for “named elements” and user defined coordinate systems.

Named elements can be curves or surfaces (see SolidWorks® user’s documentation). In the figures below you can see an example of a named element “LAGERSITZ_1” and an example of a user defined coordinate system.





If any of these entities (named elements or user defined coordinate systems) are found, the MESHPARTS software will write related information to a *.int file. Importing this information into Ansys® MAPDL has to be explicitly requested in the macro file by the following command:

```
~eui, meshparts::DefineInterfaces
```

You should place this command in the DEFINECOMPONENTS sub-macro.

The MESHPARTS software will then automatically define corresponding line and area sets (CM command) after the geometry import and meshing. Furthermore, a finite element node for all user-defined coordinate systems (excepting "Export") available in the CAD model will be defined. The new node will match the location and orientation (nodal system) of the coordinate system from the CAD model. Additionally the name of the coordinate system from the CAD model is translated into a nodal set (CM command) with the same name.

When calling the DefineInterfaces function you can specify two numerical tolerances that can be used when identifying defined interfaces based on their location and size respectively. You do that by appending the absolute tolerances to the DefineInterfaces function:

```
~eui, meshparts::DefineInterfaces 1e-3 1e-4
```

In the example above the absolute location tolerance is 1e-3 and the size tolerance is 1e-4 in model units.

20 Design of experiments

In many case you want to analyze a large number of similar FE assemblies, by varying some assembly parameters in a specified range. This type of analysis is called "Design Of Experiments" or DOE.

20.1 Generating multiple designs

In Meshparts select the main assembly in the model tree and unfold the “Design of experiments” frame:

[-] Design of experiments

Generate different designs by varying assembly parameters.
 Add one row for each design factor.
 The design factors are names of independent assembly parameters
 Enter the minimum and maximum values for each factor.
 Enter the number of equally distributed factor values in range (step
 Click "Generate table" and then "Generate models".

Factor	Min. value	Max. value	Steps
Di1	5	10	2
Di2	5	10	3

Di1	Di2
5.0	5.0
5.0	7.5
5.0	10.0
10.0	5.0
10.0	7.5
10.0	10.0

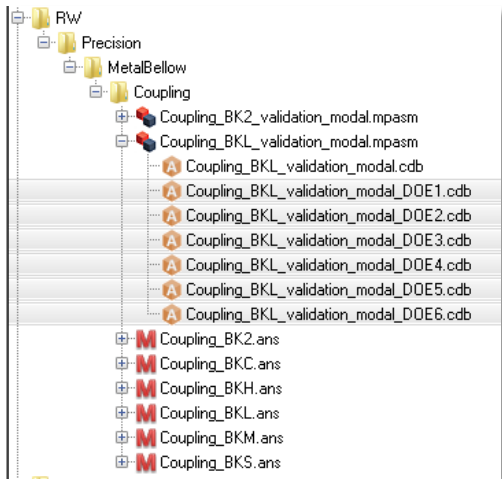
In a first step, you must add one row for each design factor. Design factors must be defined as parameters in the assembly equations frame, see Chapter 15. Specify the minimal and maximal value as real numbers. Enter the number of steps for each design factor as positive integer. The more steps you specify, the more design variants will be generated.

In the second step, click in “Generate table”. The full factorial table (all possible combinations of factor values) is automatically generated. Alternatively, you can paste your own factorial table from Excel® or a text file. Pay attention to have one column for each design factor.

If one of the factors is a string, then you must define the factorial table in Excel® or some text editor and use the “Paste” button.

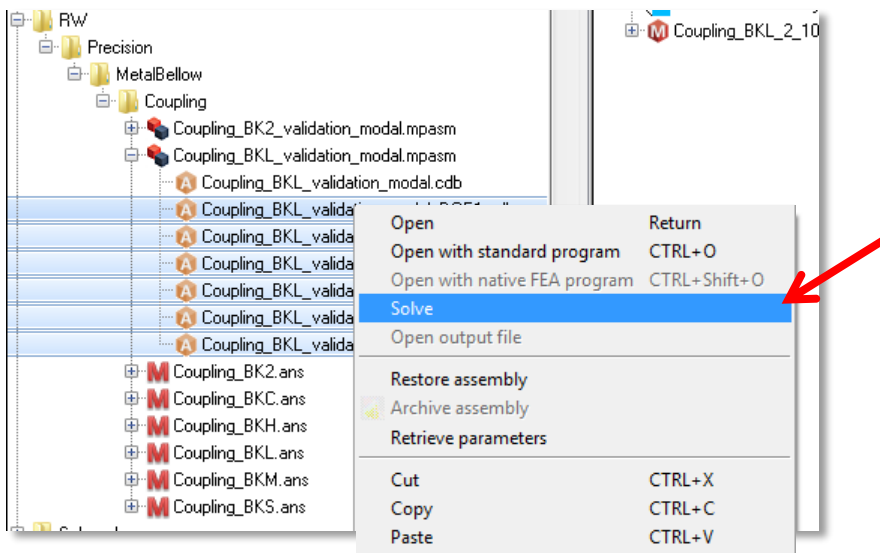
Finally, click on “Generate models” in order to generate a new assembled model for each row in the factorial table.

The new generated models are now available in the explorer tree as child elements of the assembly file. In order to differentiate between different designs, each file name ends with DOE and an index for each design:



20.2 Solving multiple designs

Now you can start the simulation of each design by selecting the assembled model files in the explorer tree and selecting from the context menu "Solve":

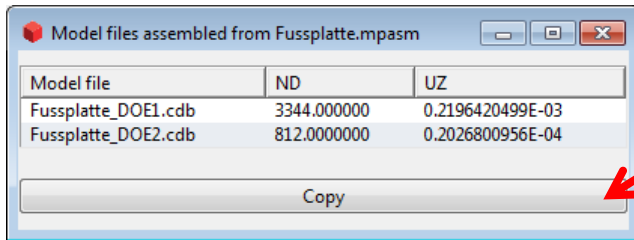


Please notice, that the type of analysis and the solution settings are the same as defined before generating the models.

Each model will be sequentially solved.

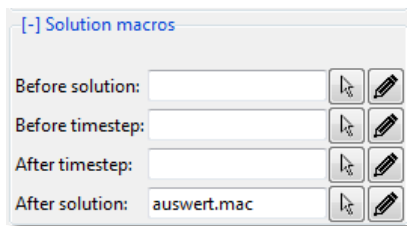
20.3 Retrieving solution parameters from designs

After all selected designs are solved, you will want to evaluate the designs based on some solution parameters that are computed using the solution macros (see Chapter 13). All these parameters and their values can be quickly retrieved from the solution output files of all designs as a table. To do this, select all exported model files and select "Retrieve parameters" from the context menu. The parameters are put into a table in a new window:



The button "Copy" will place the table content into the clipboard so that you can easily paste it into an Excel table or a text file for further processing.

Important! Only parameters that are computed using the "After solution" solution macro are taken into account:



Important! At the current stage of development, only Ansys® macros are taken into account. Automatic extraction of solution parameters is only possible for assemblies solved with Ansys®.

21 Keyboard shortcuts

Context	Keys combination	Effect
All	F1	Open user documentation
All	F11	Switch full screen mode on/off
All text fields	CTRL+A	Select all text
All context menus	Escape	Close context menu
All tables of text entries	Up or Down arrow	Jump to upper or lower entry
Explorer tree	F5	Refresh all opened explorer folders
Explorer tree	CTRL+N	Create new assembly in selected folder
Explorer tree	CTRL+SHIFT+N	Create new folder in selected folder
Explorer and model tree	CTRL+O	Open selected item with standard program
Explorer and model tree	CTRL+SHIFT+O	Open selected model with FEA program
Explorer and model tree	F2	Edit text in tree item
Explorer and model tree	Escape	Cancel current edit operation
Explorer and model tree	Arrows	Traverse tree
Explorer and model tree	Return	Open selected tree element
Explorer and model tree	Delete	Delete selected tree elements
Explorer and model tree	CTRL+C	Copy selected tree elements
Explorer and model tree	CTRL+V	Paste selected tree elements
Explorer and model tree	CTRL+X	Cut selected tree elements
Explorer and model tree	CTRL+F	Open find frame
Model tree and 3D area	CTRL+D	Duplicate selected models or assemblies
Model tree and 3D area	CTRL+E	Restore last exploded view
Model tree and 3D area	CTRL+Z	Undo last model change
Model tree and 3D area	CTRL+Y	Redo last model change
Model tree and 3D area	F5	Refresh actual model or assembly
Model tree and 3D area	1	Selection filter parts
Model tree and 3D area	2	Selection filter references
Model tree and 3D area	3	Selection filter nodal sets
Model tree and 3D area	4	Selection filter surfaces
Model tree and 3D area	5	Selection filter curves
Model tree and 3D area	6	Selection filter points
Model tree, 3D area and 3D Context menu	7	Wireframe representation
Model tree, 3D area and 3D Context menu	8	Smooth surface representation
Model tree, 3D area and 3D Context menu	9	Translucent surface representation
Model tree, 3D area and 3D Context menu	0	Transparent surface representation
3D area	Backspace	Show selected items in model tree
Find frame	Return	Start search
Find frame	Escape	Close find frame
Load step editor	x	Constrain to horizontal motion
Load step editor	y	Constrain to vertical motion
Load step editor	Escape	Cancel motion constraints
Multiple selection frame	Delete	Delete selected items in multiple selection list
Login frame text entries	Enter	Submit login form

22 Batch process execution

Some functions of the MESHPARTS software can also be executed as a batch process, meaning that no GUI will be available. Simply type the path and name of the MESHPARTS executable (e.g. C:\MESHPARTS2_20140120_x86.exe) followed by the option -b (batch). Further options and parameters can be added to the command line in order to execute a specific function. Below you can see a table of currently available command line options.

Command line option	Parameter following the option	Explanation	Parameter grouping
-b	No parameter for this option.	Signalize that the execution is done as a batch process (without GUI).	
-l	Login (e-mail address)	Your login (e-mail) is needed to obtain the software license.	Obtaining a license
-p	Password	Your password is needed to validate your e-mail.	
-e	No parameter for this option.	Signalize that an MESHPARTS assembly should be exported to a model file.	Exporting a new model from assembly
-asmpath	MESHPARTS assembly file path	The path of a MESHPARTS assembly to be exported to a model file.	
-exportpath	Path of a model file	The path of a model file to be exported from an MESHPARTS assembly.	
-g	No parameter for this option.	Signalize that a macro file should be executed in order to generate a new model.	Generating new model from macro
-path	Path of a macro file	The path of a macro file to be executed.	
-puser	List of parameter values	A list containing all macro parameter values. Input format is: "1,valu 150 2,valu 90 3,valu 8"	
-vrml	VRML export option	If this option is 1 then a VRML model is also exported.	
-iges	IGES export option	If this option is 1 then an IGES model is also exported.	
-gflag	Model regenerate option	If this option is 1 then available models will be regenerated (old models are overwritten).	

22.1 Example

The following command line runs the MESHPARTS software as a batch process and exports the assembly file myassembly.mpasm to the Ansys® model mymodel.cdb.

```
C:\MESHPARTS2_20140120_x86.exe -b -l me@domain.com -p MyPassword -asmpath C:\myassembly.mpasm -exportpath C:\mymodel.cdb
```

The following command line runs the MESHPARTS software as a batch process and generates a new Ansys® model mymacro_150_90_90_8_8_4_1.0_1_steel.cdb from the macro file mymacro.ans.

```
C:\MESHPARTS2_20140120_x86.exe -b -l me@domain.com -p MyPassword -  
path C:\mymacro.mac -puser "1,valu 150 2,valu 90 3,valu 90 4,valu 8  
5,valu 8 6,valu 4 7,valu 1.0 8,valu 1 9,valu 'steel'" -vrml 0 -iges  
0 -gflag 1
```

23 Licensing

From the first time you run the software and login with your account a trial period of one month is started. After that, the software still starts but you will not be able to open any finite element models and assemblies or generate new models.

You can also buy a commercial or academic license. Commercial and academic licenses include software updates, service and maintenance for one year and free access to most of the FE model in the online library. After that, the commercial or academic licenses expire and no new models can be generated or downloaded from the online library. You will still be able to use the software without any limitations with your local FE models. Software updates can be downloaded only with a valid license.

For more information and prices, please go to www.meshparts.de/order_license